GNU Astronomy Utilities

Astronomical data manipulation and analysis programs and libraries
for version 0.17.44-ad2d9, 3 June 2022

Important note:

This is an under-development Gnuastro release (bleeding-edge!).
It is not yet officially released.
The source tarball corresponding to this version is (temporarily) available at this URL:
http://akhlaghi.org/src/gnuastro-0.17.44-ad2d9.tar.lz
(the tarball link above will not be available after the next official release)
The most recent under-development source and its corresponding book are available at:
http://akhlaghi.org/gnuastro.pdf
http://akhlaghi.org/gnuastro-latest.tar.lz
To stay up to date with Gnuastro’s official releases, please subscribe to this mailing list:
https://lists.gnu.org/mailman/listinfo/info-gnuastro

Mohammad Akhlaghi
Gnuastro (source code, book and web page) authors (sorted by number of commits):

Mohammad Akhlaghi (mohammad@akhlaghi.org, 1944)
Raul Infante-Sainz (infantesainz@gmail.com, 97)
Pedram Ashtorofte Ardakani (pedramardakani@pm.me, 65)
Sepideh Eskandarlou (sepideh.eskandarlou@gmail.com, 37)
Mosè Giordano (mose@gnu.org, 29)
Vladimir Markelov (vmatroskin@gmail.com, 20)
Sachin Kumar Singh (sachinkumarsingh092@gmail.com, 18)
Zahra Sharbaf (zahra.sharbaf2@gmail.com, 12)
Natáli D. Anzanello (natali.anzanello@ufrgs.br, 8)
Boud Roukema (boud@cosmo.torun.pl, 7)
Carlos Morales-Socorro (cmorsoc@gmail.com, 3)
Thérèse Godefroy (godef.th@free.fr, 3)
Elham Saremi (saremi_elham@yahoo.com, 2)
Joseph Putko (josephputko@gmail.com, 2)
S. Zahra Hosseini Shahisavandi (2hs.zahra@gmail.com, 2)
Samane Raji (samaneraji@protonmail.com, 2)
Alexey Dokuchaev (danfe@freebsd.org, 1)
Andreas Stieger (astieger@suse.com, 1)
François Ochsenbein (francois.ochsenbein@gmail.com, 1)
Jash Shah (jash28582@gmail.com, 1)
Kartik Ohri (kartikohri13@gmail.com, 1)
Leindert Boogaard (leindertboogaard@gmail.com, 1)
Lucas MacQuarrie (macquarrielucas@gmail.com, 1)
Madhav Bansal (madhavbansal.cse18@itblu.ac.in, 1)
Marjan Akbari (mrjakbari@gmail.com, 1)
Miguel de Val-Borro (miguel.deval@gmail.com, 1)
Nafise Sedighi (sedighinafise94@gmail.com, 1)

This book documents version 0.17.44-ad2d9 of the GNU Astronomy Utilities (Gnuastro). Gnuastro provides various programs and libraries for astronomical data manipulation and analysis.

Copyright © 2015-2022 Free Software Foundation, Inc.

Permission is granted to copy, distribute and/or modify this document under the terms of the GNU Free Documentation License, Version 1.3 or any later version published by the Free Software Foundation; with no Invariant Sections, no Front-Cover Texts, and no Back-Cover Texts. A copy of the license is included in the section entitled “GNU Free Documentation License”.
For myself, I am interested in science and in philosophy only because I want to learn something about the riddle of the world in which we live, and the riddle of man’s knowledge of that world. And I believe that only a revival of interest in these riddles can save the sciences and philosophy from narrow specialization and from an obscurantist faith in the expert’s special skill, and in his personal knowledge and authority; a faith that so well fits our ‘post-rationalist’ and ‘post-critical’ age, proudly dedicated to the destruction of the tradition of rational philosophy, and of rational thought itself.

—Karl Popper. The logic of scientific discovery. 1959.
Short Contents

1 Introduction ......................................................... 1
2 Tutorials ................................................................. 17
3 Installation ............................................................... 119
4 Common program behavior ........................................... 152
5 Data containers .......................................................... 194
6 Data manipulation ..................................................... 259
7 Data analysis ............................................................ 337
8 Modeling and fitting ................................................... 440
9 High-level calculations ............................................... 469
10 Installed scripts ....................................................... 482
11 Library ................................................................. 513
12 Developing ............................................................. 670
A Gnuastro programs list ............................................... 699
B Other useful software .................................................. 702
C GNU Free Doc. License ............................................... 706
D GNU Gen. Pub. License v3 .......................................... 714
Index: Macros, structures and functions .............................. 725
Index ................................................................. 732
Table of Contents

1 Introduction ................................................. 1
  1.1 Quick start ............................................. 1
  1.2 Gnuastro manifesto: Science and its tools ................. 2
  1.3 Your rights ............................................. 6
  1.4 Logo of Gnuastro ......................................... 7
  1.5 Naming convention ....................................... 7
  1.6 Version numbering ....................................... 7
    1.6.1 GNU Astronomy Utilities 1.0 ......................... 8
  1.7 New to GNU/Linux? ....................................... 9
    1.7.1 Command-line interface ............................. 9
  1.8 Report a bug ............................................ 11
  1.9 Suggest new feature ..................................... 13
  1.10 Announcements ......................................... 14
  1.11 Conventions ............................................ 14
  1.12 Acknowledgments ....................................... 15

2 Tutorials ..................................................... 17
  2.1 Sufi simulates a detection ............................... 18
  2.2 General program usage tutorial .......................... 29
    2.2.1 Calling Gnuastro’s programs ......................... 30
    2.2.2 Accessing documentation ........................... 30
    2.2.3 Setup and data download ............................ 31
    2.2.4 Dataset inspection and cropping .................... 32
    2.2.5 Angular coverage on the sky ......................... 34
    2.2.6 Cosmological coverage ............................... 37
    2.2.7 Building custom programs with the library ......... 38
    2.2.8 Option management and configuration files .......... 40
    2.2.9 Warping to a new pixel grid ........................ 43
    2.2.10 NoiseChisel and Multiextension FITS files ......... 44
    2.2.11 NoiseChisel optimization for detection ............ 46
    2.2.12 NoiseChisel optimization for storage ............... 51
    2.2.13 Segmentation and making a catalog ................ 52
    2.2.14 Measuring the dataset limits ....................... 54
    2.2.15 Working with catalogs (estimating colors) .......... 58
    2.2.16 Column statistics (color-magnitude diagram) ...... 63
    2.2.17 Aperture photometry ................................ 65
    2.2.18 Matching catalogs .................................. 66
    2.2.19 Finding reddest clumps and visual inspection ...... 68
    2.2.20 Writing scripts to automate the steps .............. 69
    2.2.21 Citing and acknowledging Gnuastro ................ 76
  2.3 Detecting large extended targets ........................ 76
    2.3.1 Downloading and validating input data ............... 76
    2.3.2 NoiseChisel optimization ............................ 78
2.3.3 Skewness caused by signal and its measurement ............. 83
2.3.4 Image surface brightness limit ................................ 88
2.3.5 Achieved surface brightness level .............................. 92
2.3.6 Extract clumps and objects (Segmentation) .................. 94
2.4 Building the extended PSF ..................................... 97
  2.4.1 Preparing input for extended PSF .......................... 97
  2.4.2 Saturated pixels and Segment’s clumps ..................... 98
  2.4.3 One object for the whole detection ......................... 102
  2.4.4 Building outer part of PSF .................................. 105
  2.4.5 Inner part of the PSF ....................................... 108
  2.4.6 Uniting the different PSF components ...................... 109
  2.4.7 Subtracting the PSF ......................................... 112

3 Installation .......................................................... 119
  3.1 Dependencies ..................................................... 119
    3.1.1 Mandatory dependencies ................................. 120
      3.1.1.1 GNU Scientific Library .............................. 120
      3.1.1.2 CFITSIO ............................................. 120
      3.1.1.3 WCSLIB ............................................. 121
    3.1.2 Optional dependencies .................................... 122
    3.1.3 Bootstrapping dependencies ............................... 124
    3.1.4 Dependencies from package managers ..................... 128
  3.2 Downloading the source ....................................... 131
    3.2.1 Release tarball .......................................... 131
    3.2.2 Version controlled source ............................... 132
      3.2.2.1 Bootstrapping ....................................... 133
      3.2.2.2 Synchronizing ....................................... 134
  3.3 Build and install ............................................... 136
    3.3.1 Configuring ............................................... 136
      3.3.1.1 Gnuastro configure options .......................... 137
      3.3.1.2 Installation directory ................................ 139
      3.3.1.3 Executable names ................................... 143
      3.3.1.4 Configure and build in RAM .......................... 144
    3.3.2 Separate build and source directories .................... 145
    3.3.3 Tests ....................................................... 148
    3.3.4 A4 print book ............................................. 148
    3.3.5 Known issues ............................................... 149

4 Common program behavior ........................................ 152
  4.1 Command-line .................................................. 152
    4.1.1 Arguments and options ................................... 153
      4.1.1.1 Arguments .......................................... 154
      4.1.1.2 Options ............................................. 154
    4.1.2 Common options ........................................... 156
      4.1.2.1 Input/Output options ............................... 157
      4.1.2.2 Processing options .................................. 159
      4.1.2.3 Operating mode options ............................. 162
5 Data containers ................................. 194
  5.1 Fits ................................................. 194
      5.1.1 Invoking Fits ................................. 196
          5.1.1.1 HDU information and manipulation ..... 197
          5.1.1.2 Keyword inspection and manipulation ... 201
  5.2 ConvertType ....................................... 211
      5.2.1 Recognized file formats .................... 211
      5.2.2 Color ........................................... 214
      5.2.3 Aligning images with small WCS offsets .... 215
      5.2.4 Annotations for figure in paper ............ 216
          5.2.4.1 Full script of annotations on figure ... 222
      5.2.5 Invoking ConvertType ......................... 224
  5.3 Table ................................................. 230
      5.3.1 Column arithmetic ............................ 230
      5.3.2 Operation precedence in Table ............... 235
      5.3.3 Invoking Table ................................. 238
  5.4 Query ............................................... 249
      5.4.1 Available databases ............................ 250
      5.4.2 Invoking Query ................................. 253
6 Data manipulation ........................................... 259
  6.1 Crop .................................................... 259
    6.1.1 Crop modes ....................................... 259
    6.1.2 Crop section syntax ................................ 262
    6.1.3 Blank pixels ........................................ 262
    6.1.4 Invoking Crop ........................................ 263
      6.1.4.1 Crop options .................................... 264
      6.1.4.2 Crop output ..................................... 270
      6.1.4.3 Crop known issues ................................. 271
  6.2 Arithmetic ............................................. 272
    6.2.1 Reverse polish notation .......................... 272
    6.2.2 Integer benefits and pitfalls .................... 274
    6.2.3 Arithmetic operators ................................ 276
      6.2.3.1 Basic mathematical operators .................. 277
      6.2.3.2 Trigonometric and hyperbolic operators ...... 279
      6.2.3.3 Unit conversion operators ....................... 279
      6.2.3.4 Statistical operators ............................ 281
      6.2.3.5 Stacking operators ............................... 283
      6.2.3.6 Filtering (smoothing) operators ............... 285
      6.2.3.7 Interpolation operators ........................ 287
      6.2.3.8 Dimensionality changing operators ............ 288
      6.2.3.9 Conditional operators ............................ 290
      6.2.3.10 Mathematical morphology operators ............ 292
      6.2.3.11 Bitwise operators .............................. 294
      6.2.3.12 Numerical type conversion operators .......... 296
      6.2.3.13 Adding noise operators ........................ 297
      6.2.3.14 Elliptical shape operators ..................... 299
      6.2.3.15 Building new dataset .......................... 300
      6.2.3.16 Operand storage in memory or a file .......... 300
    6.2.4 Invoking Arithmetic ............................... 302
  6.3 Convolve ................................................. 306
    6.3.1 Spatial domain convolution ....................... 306
      6.3.1.1 Convolution process ............................ 307
      6.3.1.2 Edges in the spatial domain ..................... 307
    6.3.2 Frequency domain and Fourier operations .......... 308
      6.3.2.1 Fourier series historical background .......... 309
      6.3.2.2 Circles and the complex plane ................. 311
      6.3.2.3 Fourier series .................................. 312
      6.3.2.4 Fourier transform ............................... 314
      6.3.2.5 Dirac delta and comb ............................ 315
      6.3.2.6 Convolution theorem ............................. 316
      6.3.2.7 Sampling theorem ............................... 318
      6.3.2.8 Discrete Fourier transform ..................... 320
      6.3.2.9 Fourier operations in two dimensions .......... 322
      6.3.2.10 Edges in the frequency domain ................. 323
    6.3.3 Spatial vs. Frequency domain ...................... 323
    6.3.4 Convolution kernel ............................... 324
    6.3.5 Invoking Convolve ................................. 325
6.4 Warp ........................................................ 328
  6.4.1 Warping basics ........................................ 329
  6.4.2 Merging multiple warpings ........................... 331
  6.4.3 Resampling .............................................. 331
  6.4.4 Invoking Warp .......................................... 333

7 Data analysis ............................................. 337
  7.1 Statistics .................................................... 337
    7.1.1 Histogram and Cumulative Frequency Plot ............ 337
    7.1.2 2D Histograms ........................................... 338
      7.1.2.1 2D histogram as a table ........................ 339
      7.1.2.2 2D histogram as an image .......................... 341
    7.1.3 Sigma clipping ......................................... 343
    7.1.4 Sky value ................................................. 344
      7.1.4.1 Sky value definition .............................. 345
      7.1.4.2 Sky value misconceptions .......................... 346
      7.1.4.3 Quantifying signal in a tile ...................... 347
    7.1.5 Invoking Statistics .................................... 350
  7.2 NoiseChisel .................................................. 362
    7.2.1 NoiseChisel changes after publication .............. 364
    7.2.2 Invoking NoiseChisel ................................... 365
      7.2.2.1 NoiseChisel input ................................ 367
      7.2.2.2 Detection options .................................. 370
      7.2.2.3 NoiseChisel output ................................ 378
  7.3 Segment ...................................................... 381
    7.3.1 Invoking Segment ....................................... 382
      7.3.1.1 Segment input ..................................... 383
      7.3.1.2 Segmentation options ............................... 387
      7.3.1.3 Segment output ................................... 390
  7.4 MakeCatalog ................................................ 392
    7.4.1 Detection and catalog production ..................... 393
    7.4.2 Brightness, Flux, Magnitude and Surface brightness .. 394
    7.4.3 Quantifying measurement limits ....................... 398
      7.4.3.1 Magnitude measurement error of each detection .. 399
      7.4.3.2 Surface brightness error of each detection ...... 400
      7.4.3.3 Completeness limit of each detection ............ 400
      7.4.3.4 Upper limit magnitude of each detection .......... 400
      7.4.3.5 Magnitude limit of image .......................... 402
      7.4.3.6 Surface brightness limit of image ................ 402
      7.4.3.7 Upper limit magnitude of image .................... 404
    7.4.4 Measuring elliptical parameters ...................... 404
    7.4.5 Adding new columns to MakeCatalog .................... 407
    7.4.6 Invoking MakeCatalog .................................. 408
      7.4.6.1 MakeCatalog inputs and basic settings ............ 409
      7.4.6.2 Upper-limit settings ............................... 413
      7.4.6.3 MakeCatalog measurements .......................... 415
      7.4.6.4 MakeCatalog output ................................ 427
  7.5 Match ........................................................ 429
# 8 Modeling and fitting

8.1 MakeProfiles ........................................ 440
8.1.1 Modeling basics ....................................... 440
  8.1.1.1 Defining an ellipse and ellipsoid ............... 440
  8.1.1.2 Point spread function .......................... 442
  8.1.1.3 Stars ........................................... 444
  8.1.1.4 Galaxies ........................................ 444
  8.1.1.5 Sampling from a function ....................... 444
  8.1.1.6 Oversampling ................................... 445
8.1.2 If convolving afterwards ............................ 446
8.1.3 Profile magnitude .................................... 446
8.1.4 Invoking MakeProfiles ............................... 447
  8.1.4.1 MakeProfiles catalog ............................ 448
  8.1.4.2 MakeProfiles profile settings .................. 451
  8.1.4.3 MakeProfiles output dataset .................... 457
  8.1.4.4 MakeProfiles log file .......................... 461
8.2 MakeNoise .............................................. 461
  8.2.1 Noise basics ......................................... 461
    8.2.1.1 Photon counting noise ......................... 462
    8.2.1.2 Instrumental noise ............................ 464
    8.2.1.3 Final noised pixel value ....................... 464
    8.2.1.4 Generating random numbers ..................... 464
  8.2.2 Invoking MakeNoise ................................. 466

# 9 High-level calculations

9.1 CosmicCalculator ........................................ 469
  9.1.1 Distance on a 2D curved space ........................ 469
  9.1.2 Extending distance concepts to 3D .................. 474
9.1.3 Invoking CosmicCalculator .............................. 474
  9.1.3.1 CosmicCalculator input options ................... 475
  9.1.3.2 CosmicCalculator basic cosmology calculations .... 478
  9.1.3.3 CosmicCalculator spectral line calculations ...... 480

# 10 Installed scripts

10.1 Sort FITS files by night ............................... 483
  10.1.1 Invoking astscript-sort-by-night .................. 484
10.2 Generate radial profile ................................ 486
  10.2.1 Invoking astscript-radial-profile ................. 486
10.3 SAO DS9 region files from table ...................... 492
  10.3.1 Invoking astscript-ds9-region ..................... 492
10.4 Viewing FITS file contents with DS9 or TOPCAT ....... 494
  10.4.1 Invoking astscript-fits-view ....................... 495
10.5 PSF construction and subtraction ...................... 497
  10.5.1 Overview of the PSF scripts ....................... 498
11 Library .................................................. 513

11.1 Review of library fundamentals .............................. 513
  11.1.1 Headers .............................................. 514
  11.1.2 Linking ................................................ 517
  11.1.3 Summary and example on libraries ......................... 520

11.2 BuildProgram ............................................. 521
  11.2.1 Invoking BuildProgram .................................. 522

11.3 Gnuastro library ............................................. 525
  11.3.1 Configuration information (config.h) .................. 526
  11.3.2 Multithreaded programming (threads.h) ................. 527
    11.3.2.1 Implementation of pthread_barrier ................. 528
    11.3.2.2 Gnuastro’s thread related functions ............... 529
  11.3.3 Library data types (type.h) .......................... 532
  11.3.4 Pointers (pointer.h) .................................. 537
  11.3.5 Library blank values (blank.h) ......................... 538
  11.3.6 Data container (data.h) ................................ 542
    11.3.6.1 Generic data container (gal_data_t) ................ 542
    11.3.6.2 Dataset allocation .................................. 547
    11.3.6.3 Arrays of datasets .................................. 548
    11.3.6.4 Copying datasets ..................................... 549
  11.3.7 Dimensions (dimension.h) ................................ 550
  11.3.8 Linked lists (list.h) ................................... 554
    11.3.8.1 List of strings ...................................... 555
    11.3.8.2 List of int32_t ...................................... 556
    11.3.8.3 List of size_t ....................................... 558
    11.3.8.4 List of float ........................................ 560
    11.3.8.5 List of double ....................................... 561
    11.3.8.6 List of void * ....................................... 562
    11.3.8.7 Ordered list of size_t ............................... 564
    11.3.8.8 Doubly linked ordered list of size_t ............... 564
    11.3.8.9 List of gal_data_t ................................... 566
  11.3.9 Array input output ...................................... 567
  11.3.10 Table input output (table.h) ......................... 568
  11.3.11 FITS files (fits.h) .................................... 572
    11.3.11.1 FITS Macros, errors and filenames ................. 573
    11.3.11.2 CFITSIO and Gnuastro types ....................... 574
    11.3.11.3 FITS HDUs .......................................... 574
    11.3.11.4 FITS header keywords ................................ 576
    11.3.11.5 FITS arrays (images) ............................... 583
    11.3.11.6 FITS tables ......................................... 585
  11.3.12 File input output ...................................... 586
    11.3.12.1 Text files (txt.h) .................................. 587
11.3.12.2 TIFF files (tiff.h) .................................. 590
11.3.12.3 JPEG files (jpeg.h) .................................. 590
11.3.12.4 EPS files (eps.h) .................................. 591
11.3.12.5 PDF files (pdf.h) .................................. 592
11.3.13 World Coordinate System (wcs.h) .................... 593
11.3.14 Arithmetic on datasets (arithmetic.h) .................. 600
11.3.15 Tessellation library (tile.h) ......................... 608
   11.3.15.1 Independent tiles ................................ 609
   11.3.15.2 Tile grid ........................................ 614
11.3.16 Bounding box (box.h) ................................ 618
11.3.17 Polygons (polygon.h) ................................ 620
11.3.18 Qsort functions (qsort.h) .............................. 623
11.3.19 K-d tree (kdtree.h) ................................ 625
11.3.20 Permutations (permutation.h) .......................... 630
11.3.21 Matching (match.h) ................................ 630
11.3.22 Statistical operations (statistics.h) ................. 633
11.3.23 Binary datasets (binary.h) ......................... 640
11.3.24 Labeled datasets (label.h) ........................... 644
11.3.25 Convolution functions (convolve.h) ................. 649
11.3.26 Interpolation (interpolate.h) ........................ 650
11.3.27 Git wrappers (git.h) ................................ 654
11.3.28 Unit conversion library (units.h) .................... 655
11.3.29 Spectral lines library (speclines.h) ................. 657
11.3.30 Cosmology library (cosmology.h) ..................... 660
11.3.31 SAO DS9 library (ds9.h) ...................... 661
11.4 Library demo programs .................................. 662
   11.4.1 Library demo - reading a FITS image ................. 662
   11.4.2 Library demo - inspecting neighbors ................ 663
   11.4.3 Library demo - multi-threaded operation ............ 664
   11.4.4 Library demo - reading and writing table columns ... 667

12 Developing ............................................ 670
   12.1 Why C programming language? .......................... 670
   12.2 Program design philosophy ............................ 672
   12.3 Coding conventions .................................... 673
   12.4 Program source ....................................... 677
      12.4.1 Mandatory source code files .................... 677
      12.4.2 The TEMPLATE program ......................... 680
   12.5 Documentation ......................................... 682
   12.6 Building and debugging .............................. 683
   12.7 Test scripts .......................................... 684
   12.8 Bash programmable completion ........................ 685
      12.8.1 Bash TAB completion tutorial .................. 685
      12.8.2 Implementing TAB completion in Gnuastro ....... 689
   12.9 Developer’s checklist ................................ 690
   12.10 Gnuastro project webpage ............................ 691
   12.11 Developing mailing lists ............................ 692
   12.12 Contributing to Gnuastro ............................ 693
Appendix A  Gnuastro programs list.......... 699

Appendix B  Other useful software............ 702
   B.1 SAO DS9........................................... 702
   B.2 TOPCAT............................................. 703
   B.3 PGPLOT............................................. 703

Appendix C  GNU Free Doc. License......... 706

Appendix D  GNU Gen. Pub. License v3..... 714

Index: Macros, structures and functions ...... 725

Index ..................................................... 732
GNU Astronomy Utilities (Gnuastro) is an official GNU package consisting of separate programs and libraries for the manipulation and analysis of astronomical data. All the programs share the same basic command-line user interface for the comfort of both the users and developers. Gnuastro is written to comply fully with the GNU coding standards so it integrates finely with the GNU/Linux operating system. This also enables astronomers to expect a fully familiar experience in the source code, building, installing and command-line user interaction that they have seen in all the other GNU software that they use. The official and always up to date version of this book (or manual) is freely available under Appendix C [GNU Free Doc. License], page 706, in various formats (PDF, HTML, plain text, info, and as its Texinfo source) at http://www.gnu.org/software/gnuastro/manual/. For users who are new to the GNU/Linux environment, unless otherwise specified most of the topics in Chapter 3 [Installation], page 119, and Chapter 4 [Common program behavior], page 152, are common to all GNU software, for example installation, managing command-line options or getting help (also see Section 1.7 [New to GNU/Linux?], page 9). So if you are new to this empowering environment, we encourage you to go through these chapters carefully. They can be a starting point from which you can continue to learn more from each program's own manual and fully benefit from and enjoy this wonderful environment. Gnuastro also comes with a large set of libraries, so you can write your own programs using Gnuastro's building blocks, see Section 11.1 [Review of library fundamentals], page 513, for an introduction.

In Gnuastro, no change to any program or library will be committed to its history, before it has been fully documented here first. As discussed in Section 1.2 [Gnuastro manifesto: Science and its tools], page 2, this is a founding principle of the Gnuastro.

1.1 Quick start
The latest official release tarball is always available as gnuastro-latest.tar.gz (http://ftp.gnu.org/gnu/gnuastro/gnuastro-latest.tar.gz). For better compression (faster download), and robust archival features, an Lzip (http://www.nongnu.org/lzip/lzip.html) compressed tarball is also available at gnuastro-latest.tar.lz (http://ftp.gnu.org/gnu/gnuastro/gnuastro-latest.tar.lz), see Section 3.2.1 [Release tarball], page 131, for more details on the tarball release.

Let’s assume the downloaded tarball is in the TOPGNUASTRO directory. The first two commands below can be used to decompress the source. If you download tar.lz and your Tar implementation doesn’t recognize Lzip (the second command fails), run the third and fourth lines. Note that lines starting with `##` don’t need to be typed.

```
## Go into the download directory.
```

1 The Gzip library and program are commonly available on most systems. However, Gnuastro recommends Lzip as described above and the beta-releases are also only distributed in tar.lz. You can download and install Lzip’s source (in .tar.gz format) from its web page and follow the same process as below: Lzip has no dependencies, so simply decompress, then run ./configure, make, sudo make install.

2 In case Tar doesn’t directly uncompress your .tar.lz tarball, you can merge the separate calls to Lzip and Tar (shown in the main body of text) into one command by directly piping the output of Lzip into Tar with a command like this: `$ lzip -cd gnuastro-0.5.tar.lz | tar -xf`.
Chapter 1: Introduction

$ cd TOPGNUASTRO

## Also works on ‘tar.gz’. GNU Tar recognizes both formats.
$ tar xf gnuastro-latest.tar.lz

## Only when the previous command fails.
$ lzip -d gnuastro-latest.tar.lz
$ tar xf gnuastro-latest.tar

Gnuastro has three mandatory dependencies and some optional dependencies for extra functionality, see Section 3.1 [Dependencies], page 119, for the full list. In Section 3.1.4 [Dependencies from package managers], page 128, we have prepared the command to easily install Gnuastro’s dependencies using the package manager of some operating systems. When the mandatory dependencies are ready, you can configure, compile, check and install Gnuastro on your system with the following commands. See Section 3.3.5 [Known issues], page 149, if you confront any complications.

$ cd gnuastro-X.X # Replace X.X with version number.
$ ./configure
$ make -j8 # Replace 8 with no. CPU threads.
$ make check -j8 # Replace 8 with no. CPU threads.
$ sudo make install
$ echo "source /usr/local/share/gnuastro/completion.bash" >> ~/.bashrc

The last command is to enable Gnuastro’s custom TAB completion in Bash. For more on this useful feature, see Section 4.1.3 [Shell TAB completion (highly customized)], page 166.

For each program there is an ‘Invoke ProgramName’ sub-section in this book which explains how the programs should be run on the command-line (for example Section 5.3.3 [Invoking Table], page 238).

Some complete Tutorials are also available in this book with common Gnuastro usage scenarios in astronomical research. They even contain links to download the necessary data, and thoroughly describe every step of the process (the science, statistics and optimal usage of the command-line). We therefore strongly recommend to follow the tutorials before starting to use Gnuastro, see Chapter 2 [Tutorials], page 17.

1.2 Gnuastro manifesto: Science and its tools

History of science indicates that there are always inevitably unseen faults, hidden assumptions, simplifications and approximations in all our theoretical models, data acquisition and analysis techniques. It is precisely these that will ultimately allow future generations to advance the existing experimental and theoretical knowledge through their new solutions and corrections.

In the past, scientists would gather data and process them individually to achieve an analysis thus having a much more intricate knowledge of the data and analysis. The theoretical models also required little (if any) simulations to compare with the data. Today both methods are becoming increasingly more dependent on pre-written software. Scientists are dissociating themselves from the intricacies of reducing raw observational data in experimentation or from bringing the theoretical models to life in simulations. These ‘intricacies’
are precisely those unseen faults, hidden assumptions, simplifications and approximations that define scientific progress.

Unfortunately, most persons who have recourse to a computer for statistical analysis of data are not much interested either in computer programming or in statistical method, being primarily concerned with their own proper business. Hence the common use of library programs and various statistical packages. ... It's time that was changed.


Anscombe’s quartet (http://en.wikipedia.org/wiki/Anscombe%27s_quartet) demonstrates how four data sets with widely different shapes (when plotted) give nearly identical output from standard regression techniques. Anscombe uses this (now famous) quartet, which was introduced in the paper quoted above, to argue that “Good statistical analysis is not a purely routine matter, and generally calls for more than one pass through the computer”. Echoing Anscombe’s concern after 44 years, some of the highly recognized statisticians of our time (Leek, McShane, Gelman, Colquhoun, Nuijten and Goodman), wrote in Nature that:

We need to appreciate that data analysis is not purely computational and algorithmic – it is a human behaviour....Researchers who hunt hard enough will turn up a result that fits statistical criteria – but their discovery will probably be a false positive.


Users of statistical (scientific) methods (software) are therefore not passive (objective) agents in their results. It is necessary to actually understand the method, not just use it as a black box. The subjective experience gained by frequently using a method/software is not sufficient to claim an understanding of how the tool/method works and how relevant it is to the data and analysis. This kind of subjective experience is prone to serious misunderstandings about the data, what the software/statistical-method really does (especially as it gets more complicated), and thus the scientific interpretation of the result. This attitude is further encouraged through non-free software³, poorly written (or non-existent) scientific software manuals, and non-reproducible papers⁴. This approach to scientific software and methods only helps in producing dogmas and an “obscurantist faith in the expert’s special skill, and in his personal knowledge and authority”⁵.

Program or be programmed. Choose the former, and you gain access to the control panel of civilization. Choose the latter, and it could be the last real choice you get to make.


It is obviously impractical for any one human being to gain the intricate knowledge explained above for every step of an analysis. On the other hand, scientific data can

³ https://www.gnu.org/philosophy/free-sw.html

⁴ Where the authors omit many of the analysis/processing “details” from the paper by arguing that they would make the paper too long/unreadable. However, software engineers have been dealing with such issues for a long time. There are thus software management solutions that allow us to supplement papers with all the details necessary to exactly reproduce the result. For example see Akhlaghi et al. (2021, arXiv:2006.03018 (https://arxiv.org/abs/2006.03018)).

⁵ Karl Popper. The logic of scientific discovery. 1959. Larger quote is given at the start of the PDF (for print) version of this book.
be large and numerous, for example images produced by telescopes in astronomy. This requires efficient algorithms. To make things worse, natural scientists have generally not been trained in the advanced software techniques, paradigms and architecture that are taught in computer science or engineering courses and thus used in most software. The GNU Astronomy Utilities are an effort to tackle this issue.

Gnuastro is not just a software, this book is as important to the idea behind Gnuastro as the source code (software). This book has tried to learn from the success of the “Numerical Recipes” book in educating those who are not software engineers and computer scientists but still heavy users of computational algorithms, like astronomers. There are two major differences.

The first difference is that Gnuastro’s code and the background information are segregated: the code is moved within the actual Gnuastro software source code and the underlying explanations are given here in this book. In the source code, every non-trivial step is heavily commented and correlated with this book, it follows the same logic of this book, and all the programs follow a similar internal data, function and file structure, see Section 12.4 [Program source], page 677. Complementing the code, this book focuses on thoroughly explaining the concepts behind those codes (history, mathematics, science, software and usage advice when necessary) along with detailed instructions on how to run the programs. At the expense of frustrating “professionals” or “experts”, this book and the comments in the code also intentionally avoid jargon and abbreviations. The source code and this book are thus intimately linked, and when considered as a single entity can be thought of as a real (an actual software accompanying the algorithms) “Numerical Recipes” for astronomy.

The second major, and arguably more important, difference is that “Numerical Recipes” does not allow you to distribute any code that you have learned from it. In other words, it does not allow you to release your software’s source code if you have used their codes, you can only publicly release binaries (a black box) to the community. Therefore, while it empowers the privileged individual who has access to it, it exacerbates social ignorance. Exactly at the opposite end of the spectrum, Gnuastro’s source code is released under the GNU general public license (GPL) and this book is released under the GNU free documentation license. You are therefore free to distribute any software you create using parts of Gnuastro’s source code or text, or figures from this book, see Section 1.3 [Your rights], page 6.

With these principles in mind, Gnuastro’s developers aim to impose the minimum requirements on you (in computer science, engineering and even the mathematics behind the tools) to understand and modify any step of Gnuastro if you feel the need to do so, see Section 12.1 [Why C programming language?], page 670, and Section 12.2 [Program design philosophy], page 672.

Without prior familiarity and experience with optics, it is hard to imagine how, Galileo could have come up with the idea of modifying the Dutch military telescope optics to use in astronomy. Astronomical objects could not be seen with the Dutch military design of the telescope. In other words, it is unlikely that Galileo could have asked a random optician to make modifications (not understood by Galileo) to the Dutch design, to do something no astronomer of the time took seriously. In the paradigm of the day, what could be the purpose of enlarging geometric spheres (planets) or points (stars)? In that paradigm only the position and movement of the heavenly bodies was important, and that had already been accurately studied (recently by Tycho Brahe).
In the beginning of his "The Sidereal Messenger" (published in 1610) he cautions the readers on this issue and before describing his results/observations, Galileo instructs us on how to build a suitable instrument. Without a detailed description of how he made his tools and done his observations, no reasonable person would believe his results. Before he actually saw the moons of Jupiter, the mountains on the Moon or the crescent of Venus, Galileo was “evasive” to Kepler. Science is defined by its tools/methods, not its raw results.

The same is true today: science cannot progress with a black box, or poorly released code. The source code of a research is the new (abstractified) communication language in science, understandable by humans and computers. Source code (in any programming language) is a language/notation designed to express all the details that would be too tedious/long/frustrating to report in spoken languages like English, similar to mathematic notation.

An article about computational science [almost all sciences today] ... is not the scholarship itself, it is merely advertising of the scholarship. The Actual Scholarship is the complete software development environment and the complete set of instructions which generated the figures.


Today, the quality of the source code that goes into a scientific result (and the distribution of that code) is as critical to scientific vitality and integrity, as the quality of its written language/English used in publishing/distributing its paper. A scientific paper will not even be reviewed by any respectable journal if its written in a poor language/English. A similar level of quality assessment is thus increasingly becoming necessary regarding the codes/methods used to derive the results of a scientific paper. For more on this, please see Akhlaghi et al. (2021) at arXiv:2006.03018 (https://arxiv.org/abs/2006.03018).

Bjarne Stroustrup (creator of the C++ language) says: "Without understanding software, you are reduced to believing in magic". Ken Thomson (the designer of the Unix operating system) says “I abhor a system designed for the ‘user’ if that word is a coded pejorative meaning ‘stupid and unsophisticated’.” Certainly no scientist (user of a scientific software) would want to be considered a believer in magic, or stupid and unsophisticated.

This can happen when scientists get too distant from the raw data and methods, and are mainly discussing results. In other words, when they feel they have tamed Nature into their own high-level (abstract) models (creations), and are mainly concerned with scaling up, or industrializing those results. Roughly five years before special relativity, and about two decades before quantum mechanics fundamentally changed Physics, Lord Kelvin is quoted as saying:

There is nothing new to be discovered in physics now. All that remains is more and more precise measurement.

—William Thomson (Lord Kelvin), 1900

A few years earlier Albert. A. Michelson made the following statement:

---


7 For example, take the following two results on the age of the universe: roughly 14 billion years (suggested by the current consensus of the standard model of cosmology) and less than 10,000 years (suggested from some interpretations of the Bible). Both these numbers are results. What distinguishes these two results, is the tools/methods that were used to derive them. Therefore, as the term “Scientific method” also signifies, a scientific statement it defined by its method, not its result.
The more important fundamental laws and facts of physical science have all been discovered, and these are now so firmly established that the possibility of their ever being supplanted in consequence of new discoveries is exceedingly remote. Our future discoveries must be looked for in the sixth place of decimals.
—Albert. A. Michelson, dedication of Ryerson Physics Lab, U. Chicago 1894

If scientists are considered to be more than mere puzzle solvers (simply adding to the decimals of existing values or observing a feature in 10, 100, or 100000 more galaxies or stars, as Kelvin and Michelson clearly believed), they cannot just passively sit back and uncritically repeat the previous (observational or theoretical) methods/tools on new data. Today there is a wealth of raw telescope images ready (mostly for free) at the finger tips of anyone who is interested with a fast enough internet connection to download them. The only thing lacking is new ways to analyze this data and dig out the treasure that is lying hidden in them to existing methods and techniques.

New data that we insist on analyzing in terms of old ideas (that is, old models which are not questioned) cannot lead us out of the old ideas. However many data we record and analyze, we may just keep repeating the same old errors, missing the same crucially important things that the experiment was competent to find.

1.3 Your rights
The paragraphs below, in this section, belong to the GNU Texinfo manual and are not written by us! The name “Texinfo” is just changed to “GNU Astronomy Utilities” or “Gnuastro” because they are released under the same licenses and it is beautifully written to inform you of your rights.

GNU Astronomy Utilities is “free software”; this means that everyone is free to use it and free to redistribute it on certain conditions. Gnuastro is not in the public domain; it is copyrighted and there are restrictions on its distribution, but these restrictions are designed to permit everything that a good cooperating citizen would want to do. What is not allowed is to try to prevent others from further sharing any version of Gnuastro that they might get from you.

Specifically, we want to make sure that you have the right to give away copies of the programs that relate to Gnuastro, that you receive the source code or else can get it if you want it, that you can change these programs or use pieces of them in new free programs, and that you know you can do these things.

To make sure that everyone has such rights, we have to forbid you to deprive anyone else of these rights. For example, if you distribute copies of the Gnuastro related programs, you must give the recipients all the rights that you have. You must make sure that they, too, receive or can get the source code. And you must tell them their rights.

Also, for our own protection, we must make certain that everyone finds out that there is no warranty for the programs that relate to Gnuastro. If these programs are modified by someone else and passed on, we want their recipients to know that what they have is

---

9 Texinfo is the GNU documentation system. It is used to create this book in all the various formats.
not what we distributed, so that any problems introduced by others will not reflect on our reputation.

The full text of the licenses for the Gnuastro book and software can be respectively found in Appendix D [GNU Gen. Pub. License v3], page 714\(^{10}\) and Appendix C [GNU Free Doc. License], page 706\(^{11}\).

1.4 Logo of Gnuastro

Gnuastro’s logo is an abstract image of a barred spiral galaxy (https://en.wikipedia.org/wiki/Barred_spiral_galaxy). The galaxy is vertically cut in half: on the left side, the beauty of a contiguous galaxy image is visible. But on the right, the image gets pixelated, and we only see the parts that are within the pixels. The pixels that are more near to the center of the galaxy (which is brighter) are also larger. But as we follow the spiral arms (and get more distant from the center), the pixels get smaller (signifying less signal).

This sharp distinction between the contiguous and pixelated view of the galaxy signifies the main struggle in science: in the “real” world, objects aren’t pixelated or discrete and have no noise. However, when we observe nature, we are confined and constrained by the resolution of our data collection (CCD imager in this case).

On the other hand, we read English text from the left and progress towards the right. This defines the positioning of the “real” and observed halves of the galaxy: the no-noised and contiguous half (on the left) passes through our observing tools and becomes pixelated and noisy half (on the right). It is the job of scientific software like Gnuastro to help interpret the underlying mechanisms of the “real” universe from the pixelated and noisy data.

Gnuastro’s logo was designed by Marjan Akbari. The concept behind it was created after several design iterations with Mohammad Akhlaghi.

1.5 Naming convention

Gnuastro is a package of independent programs and a collection of libraries, here we are mainly concerned with the programs. Each program has an official name which consists of one or two words, describing what they do. The latter are printed with no space, for example NoiseChisel or Crop. On the command-line, you can run them with their executable names which start with an `ast` and might be an abbreviation of the official name, for example `astnoisechisel` or `astcrop`, see Section 3.3.1.3 [Executable names], page 143.

We will use “ProgramName” for a generic official program name and `astprogname` for a generic executable name. In this book, the programs are classified based on what they do and thoroughly explained. An alphabetical list of the programs that are installed on your system with this installation are given in Appendix A [Gnuastro programs list], page 699. That list also contains the executable names and version numbers along with a one line description.

1.6 Version numbering

Gnuastro can have two formats of version numbers, for official and unofficial releases. Official Gnuastro releases are announced on the `info-gnuastro` mailing list, they have a version

\(^{10}\) Also available in http://www.gnu.org/copyleft/gpl.html

\(^{11}\) Also available in http://www.gnu.org/copyleft/fdl.html
control tag in Gnuastro’s development history, and their version numbers are formatted like “A.B”. \( A \) is a major version number, marking a significant planned achievement (for example see Section 1.6.1 [GNU Astronomy Utilities 1.0], page 8), while \( B \) is a minor version number, see below for more on the distinction. Note that the numbers are not decimals, so version 2.34 is much more recent than version 2.5, which is not equal to 2.50.

Gnuastro also allows a unique version number for unofficial releases. Unofficial releases can mark any point in Gnuastro’s development history. This is done to allow astronomers to easily use any point in the version controlled history for their data-analysis and research publication. See Section 3.2.2 [Version controlled source], page 132, for a complete introduction. This section is not just for developers and is intended to straightforward and easy to read, so please have a look if you are interested in the cutting-edge. This unofficial version number is a meaningful and easy to read string of characters, unique to that particular point of history. With this feature, users can easily stay up to date with the most recent bug fixes and additions that are committed between official releases.

The unofficial version number is formatted like: \( A.B.C-D \). \( A \) and \( B \) are the most recent official version number. \( C \) is the number of commits that have been made after version \( A.B \). \( D \) is the first 4 or 5 characters of the commit hash number\(^\text{12}\). Therefore, the unofficial version number ‘3.92.8-29c8’, corresponds to the 8th commit after the official version 3.92 and its commit hash begins with 29c8. The unofficial version number is sort-able (unlike the raw hash) and as shown above is descriptive of the state of the unofficial release. Of course an official release is preferred for publication (since its tarballs are easily available and it has gone through more tests, making it more stable), so if an official release is announced prior to your publication’s final review, please consider updating to the official release.

The major version number is set by a major goal which is defined by the developers and user community before hand, for example see Section 1.6.1 [GNU Astronomy Utilities 1.0], page 8. The incremental work done in minor releases are commonly small steps in achieving the major goal. Therefore, there is no limit on the number of minor releases and the difference between the (hypothetical) versions 2.927 and 3.0 can be a small (negligible to the user) improvement that finalizes the defined goals.

### 1.6.1 GNU Astronomy Utilities 1.0

Currently (prior to Gnuastro 1.0), the aim of Gnuastro is to have a complete system for data manipulation and analysis at least similar to IRAF\(^\text{13}\). So an astronomer can take all the standard data analysis steps (starting from raw data to the final reduced product and standard post-reduction tools) with the various programs in Gnuastro.

The maintainers of each camera or detector on a telescope can provide a completely transparent shell script or Makefile to the observer for data analysis. This script can set configuration files for all the required programs to work with that particular camera. The script can then run the proper programs in the proper sequence. The user/observer can easily follow the standard shell script to understand (and modify) each step and the parameters used easily. Bash (or other modern GNU/Linux shell scripts) is powerful and made

\(^{12}\) Each point in Gnuastro’s history is uniquely identified with a 40 character long hash which is created from its contents and previous history for example: 5b17501d8f29ba3cd610673261e6e2229c846d35. So the string \( D \) in the version for this commit could be 5b17, or 5b175.

\(^{13}\) http://iraf.noao.edu/
for this gluing job. This will simultaneously improve performance and transparency. Shell scripting (or Makefiles) are also basic constructs that are easy to learn and readily available as part of the Unix-like operating systems. If there is no program to do a desired step, Gnuastro’s libraries can be used to build specific programs.

The main factor is that all observatories or projects can freely contribute to Gnuastro and all simultaneously benefit from it (since it doesn’t belong to any particular one of them), much like how for-profit organizations (for example RedHat, or Intel and many others) are major contributors to free and open source software for their shared benefit. Gnuastro’s copyright has been fully awarded to GNU, so it doesn’t belong to any particular astronomer or astronomical facility or project.

1.7 New to GNU/Linux?

Some astronomers initially install and use a GNU/Linux operating system because their necessary tools can only be installed in this environment. However, the transition is not necessarily easy. To encourage you in investing the patience and time to make this transition, and actually enjoy it, we will first start with a basic introduction to GNU/Linux operating systems. Afterwards, in Section 1.7.1 [Command-line interface], page 9, we’ll discuss the wonderful benefits of the command-line interface, how it beautifully complements the graphic user interface, and why it is worth the (apparently steep) learning curve. Finally a complete chapter (Chapter 2 [Tutorials], page 17) is devoted to real world scenarios of using Gnuastro (on the command-line). Therefore if you don’t yet feel comfortable with the command-line we strongly recommend going through that chapter after finishing this section.

You might have already noticed that we are not using the name “Linux”, but “GNU/Linux”. Please take the time to have a look at the following essays and FAQs for a complete understanding of this very important distinction.


In short, the Linux kernel is built using the GNU C library (glibc) and GNU compiler collection (gcc). The Linux kernel software alone is just a means for other software to access the hardware resources, it is useless alone: to say “running Linux”, is like saying “driving your carburetor”.

To have an operating system, you need lower-level (to build the kernel), and higher-level (to use it) software packages. The majority of such software in most Unix-like operating systems are GNU software: “the whole system is basically GNU with Linux loaded”. Therefore to acknowledge GNU’s instrumental role in the creation and usage of the Linux kernel and the operating systems that use it, we should call these operating systems “GNU/Linux”.

1.7.1 Command-line interface

One aspect of Gnuastro that might be a little troubling to new GNU/Linux users is that (at least for the time being) it only has a command-line user interface (CLI). This might be

\[14\] In Unix-like operating systems, the kernel connects software and hardware worlds.
contrary to the mostly graphical user interface (GUI) experience with proprietary operating systems. Since the various actions available aren’t always on the screen, the command-line interface can be complicated, intimidating, and frustrating for a first-time user. This is understandable and also experienced by anyone who started using the computer (from childhood) in a graphical user interface (this includes most of Gnuastro’s authors). Here we hope to convince you of the unique benefits of this interface which can greatly enhance your productivity while complementing your GUI experience.

Through GNOME 315, most GNU/Linux based operating systems now have an advanced and useful GUI. Since the GUI was created long after the command-line, some wrongly consider the command line to be obsolete. Both interfaces are useful for different tasks. For example you can’t view an image, video, pdf document or web page on the command-line. On the other hand you can’t reproduce your results easily in the GUI. Therefore they should not be regarded as rivals but as complementary user interfaces, here we will outline how the CLI can be useful in scientific programs.

You can think of the GUI as a veneer over the CLI to facilitate a small subset of all the possible CLI operations. Each click you do on the GUI, can be thought of as internally running a different CLI command. So asymptotically (if a good designer can design a GUI which is able to show you all the possibilities to click on) the GUI is only as powerful as the command-line. In practice, such graphical designers are very hard to find for every program, so the GUI operations are always a subset of the internal CLI commands. For programs that are only made for the GUI, this results in not including lots of potentially useful operations. It also results in ‘interface design’ to be a crucially important part of any GUI program. Scientists don’t usually have enough resources to hire a graphical designer, also the complexity of the GUI code is far more than CLI code, which is harmful for a scientific software, see Section 1.2 [Gnuastro manifesto: Science and its tools], page 2.

For programs that have a GUI, one action on the GUI (moving and clicking a mouse, or tapping a touchscreen) might be more efficient and easier than its CLI counterpart (typing the program name and your desired configuration). However, if you have to repeat that same action more than once, the GUI will soon become frustrating and prone to errors. Unless the designers of a particular program decided to design such a system for a particular GUI action, there is no general way to run any possible series of actions automatically on the GUI.

On the command-line, you can run any series of actions which can come from various CLI capable programs you have decided your self in any possible permutation with one command16. This allows for much more creativity and exact reproducibility that is not possible to a GUI user. For technical and scientific operations, where the same operation (using various programs) has to be done on a large set of data files, this is crucially important. It also allows exact reproducibility which is a foundation principle for scientific results. The most common CLI (which is also known as a shell) in GNU/Linux is GNU Bash, we strongly encourage you to put aside several hours and go through this beautifully explained web page: https://flossmanuals.net/command-line/. You don’t need to read or even fully understand the whole thing, only a general knowledge of the first few chapters are enough to get you going.

---

15 http://www.gnome.org/
16 By writing a shell script and running it, for example see the tutorials in Chapter 2 [Tutorials], page 17.
Since the operations in the GUI are limited and they are visible, reading a manual is not that important in the GUI (most programs don’t even have any!). However, to give you the creative power explained above, with a CLI program, it is best if you first read the manual of any program you are using. You don’t need to memorize any details, only an understanding of the generalities is needed. Once you start working, there are more easier ways to remember a particular option or operation detail, see Section 4.3 [Getting help], page 172.

To experience the command-line in its full glory and not in the GUI terminal emulator, press the following keys together: $\texttt{CTRL+ALT+F4}$ to access the virtual console. To return back to your GUI, press the same keys above replacing F4 with F7 (or F1, or F2, depending on your GNU/Linux distribution). In the virtual console, the GUI, with all its distracting colors and information, is gone. Enabling you to focus entirely on your actual work.

For operations that use a lot of your system’s resources (processing a large number of large astronomical images for example), the virtual console is the place to run them. This is because the GUI is not competing with your research work for your system’s RAM and CPU. Since the virtual consoles are completely independent, you can even log out of your GUI environment to give even more of your hardware resources to the programs you are running and thus reduce the operating time.

Since it uses far less system resources, the CLI is also convenient for remote access to your computer. Using secure shell (SSH) you can log in securely to your system (similar to the virtual console) from anywhere even if the connection speeds are low. There are apps for smart phones and tablets which allow you to do this.

### 1.8 Report a bug

According to Wikipedia “a software bug is an error, flaw, failure, or fault in a computer program or system that causes it to produce an incorrect or unexpected result, or to behave in unintended ways”. So when you see that a program is crashing, not reading your input correctly, giving the wrong results, or not writing your output correctly, you have found a bug. In such cases, it is best if you report the bug to the developers. The programs will also inform you if known impossible situations occur (which are caused by something unexpected) and will ask the users to report the bug issue.

Prior to actually filing a bug report, it is best to search previous reports. The issue might have already been found and even solved. The best place to check if your bug has already been discussed is the bugs tracker on Section 12.10 [Gnuastro project webpage], page 691, at https://savannah.gnu.org/bugs/?group=gnuastro. In the top search fields (under “Display Criteria”) set the “Open/Closed” drop-down menu to “Any” and choose the respective program or general category of the bug in “Category” and click the “Apply” button. The results colored green have already been solved and the status of those colored in red is shown in the table.

Recently corrected bugs are probably not yet publicly released because they are scheduled for the next Gnuastro stable release. If the bug is solved but not yet released and it is

---

17 Instead of $\texttt{F4}$, you can use any of the keys from $\texttt{F1}$ to $\texttt{F6}$ for different virtual consoles depending on your GNU/Linux distribution, try them all out. You can also run a separate GUI from within this console if you want to.
an urgent issue for you, you can get the version controlled source and compile that, see Section 3.2.2 [Version controlled source], page 132.

To solve the issue as readily as possible, please follow the following to guidelines in your bug report. The How to Report Bugs Effectively (http://www.chiark.greenend.org.uk/~sgtatham/bugs.html) and How To Ask Questions The Smart Way (http://catb.org/~esr/faqs/smart-questions.html) essays also provide some good generic advice for all software (don’t contact their authors for Gnuastro’s problems). Mastering the art of giving good bug reports (like asking good questions) can greatly enhance your experience with any free and open source software. So investing the time to read through these essays will greatly reduce your frustration after you see something doesn’t work the way you feel it is supposed to for a large range of software, not just Gnuastro.

Be descriptive

Please provide as many details as possible and be very descriptive. Explain what you expected and what the output was: it might be that your expectation was wrong. Also please clearly state which sections of the Gnuastro book (this book), or other references you have studied to understand the problem. This can be useful in correcting the book (adding links to likely places where users will check). But more importantly, it will be encouraging for the developers, since you are showing how serious you are about the problem and that you have actually put some thought into it. “To be able to ask a question clearly is two-thirds of the way to getting it answered.” – John Ruskin (1819-1900).

Individual and independent bug reports

If you have found multiple bugs, please send them as separate (and independent) bugs (as much as possible). This will significantly help us in managing and resolving them sooner.

Reproducible bug reports

If we cannot exactly reproduce your bug, then it is very hard to resolve it. So please send us a Minimal working example18 along with the description. For example in running a program, please send us the full command-line text and the output with the -P option, see Section 4.1.2.3 [Operating mode options], page 162. If it is caused only for a certain input, also send us that input file. In case the input FITS is large, please use Crop to only crop the problematic section and make it as small as possible so it can easily be uploaded and downloaded and not waste the archive’s storage, see Section 6.1 [Crop], page 259.

There are generally two ways to inform us of bugs:

• Send a mail to bug-gnuastro@gnu.org. Any mail you send to this address will be distributed through the bug-gnuastro mailing list19. This is the simplest way to send us bug reports. The developers will then register the bug into the project web page (next choice) for you.

• Use the Gnuastro project web page at https://savannah.gnu.org/projects/gnuastro/: There are two ways to get to the submission page as listed below. Fill

18 http://en.wikipedia.org/wiki/Minimal_Working_Example
19 https://lists.gnu.org/mailman/listinfo/bug-gnuastro
in the form as described below and submit it (see Section 12.10 [Gnuastro project webpage], page 691, for more on the project web page).

- Using the top horizontal menu items, immediately under the top page title. Hovering your mouse on “Support” will open a drop-down list. Select “Submit new”.

- In the main body of the page, under the “Communication tools” section, click on “Submit new item”.

Once the items have been registered in the mailing list or web page, the developers will add it to either the “Bug Tracker” or “Task Manager” trackers of the Gnuastro project web page. These two trackers can only be edited by the Gnuastro project developers, but they can be browsed by anyone, so you can follow the progress on your bug. You are most welcome to join us in developing Gnuastro and fixing the bug you have found maybe a good starting point. Gnuastro is designed to be easy for anyone to develop (see Section 1.2 [Gnuastro manifesto: Science and its tools], page 2) and there is a full chapter devoted to developing it: Chapter 12 [Developing], page 670.

1.9 Suggest new feature

We would always be happy to hear of suggested new features. For every program, there are already lists of features that we are planning to add. You can see the current list of plans from the Gnuastro project web page at https://savannah.gnu.org/projects/gnuastro/ and following “Tasks” → “Browse” on the horizontal menu at the top of the page immediately under the title, see Section 12.10 [Gnuastro project webpage], page 691. If you want to request a feature to an existing program, click on the “Display Criteria” above the list and under “Category”, choose that particular program. Under “Category” you can also see the existing suggestions for new programs or other cases like installation, documentation or libraries. Also, be sure to set the “Open/Closed” value to “Any”.

If the feature you want to suggest is not already listed in the task manager, then follow the steps that are fully described in Section 1.8 [Report a bug], page 11. Please have in mind that the developers are all busy with their own astronomical research, and implementing existing “task”s to add or resolve bugs. Gnuastro is a volunteer effort and none of the developers are paid for their hard work. So, although we will try our best, please don’t expect for your suggested feature to be immediately included (for the next release of Gnuastro).

The best person to apply the exciting new feature you have in mind is you, since you have the motivation and need. In fact, Gnuastro is designed for making it as easy as possible for you to hack into it (add new features, change existing ones and so on), see Section 1.2 [Gnuastro manifesto: Science and its tools], page 2. Please have a look at the chapter devoted to developing (Chapter 12 [Developing], page 670) and start applying your desired feature. Once you have added it, you can use it for your own work and if you feel you want others to benefit from your work, you can request for it to become part of Gnuastro. You can then join the developers and start maintaining your own part of Gnuastro. If you choose to take this path of action please contact us beforehand (Section 1.8 [Report a bug], page 11) so we can avoid possible duplicate activities and get interested people in contact.
Gnuastro is a collection of low level programs: As described in Section 12.2 [Program design philosophy], page 672, a founding principle of Gnuastro is that each library or program should be basic and low-level. High level jobs should be done by running the separate programs or using separate functions in succession through a shell script or calling the libraries by higher level functions, see the examples in Chapter 2 [Tutorials], page 17. So when making the suggestions please consider how your desired job can best be broken into separate steps and modularized.

1.10 Announcements

Gnuastro has a dedicated mailing list for making announcements (info-gnuastro). Anyone can subscribe to this mailing list. Anytime there is a new stable or test release, an email will be circulated there. The email contains a summary of the overall changes along with a detailed list (from the NEWS file). This mailing list is thus the best way to stay up to date with new releases, easily learn about the updated/new features, or dependencies (see Section 3.1 [Dependencies], page 119).

To subscribe to this list, please visit https://lists.gnu.org/mailman/listinfo/info-gnuastro. Traffic (number of mails per unit time) in this list is designed to be low: only a handful of mails per year. Previous announcements are available on its archive (http://lists.gnu.org/archive/html/info-gnuastro/).

1.11 Conventions

In this book we have the following conventions:

- All commands that are to be run on the shell (command-line) prompt as the user start with a $. In case they must be run as a super-user or system administrator, they will start with a single#. If the command is in a separate line and next line is also in the code type face, but doesn’t have any of the $ or # signs, then it is the output of the command after it is run. As a user, you don’t need to type those lines. A line that starts with ## is just a comment for explaining the command to a human reader and must not be typed.

- If the command becomes larger than the page width a \ is inserted in the code. If you are typing the code by hand on the command-line, you don’t need to use multiple lines or add the extra space characters, so you can omit them. If you want to copy and paste these examples (highly discouraged!) then the \ should stay.

The \ character is a shell escape character which is used commonly to make characters which have special meaning for the shell, lose that special meaning (the shell will not treat them especially if there is a \ behind them). When \ is the last visible character in a line (the next character is a new-line character) the new-line character loses its meaning. Therefore, the shell sees it as a simple white-space character not the end of a command! This enables you to use multiple lines to write your commands.

This is not a convention, but a bi-product of the PDF building process of the manual: In the PDF version of this manual, a single quote (or apostrophe) character in the commands or codes is shown like this: `. Single quotes are sometimes necessary in combination with commands like awk or sed, or when using Column arithmetic in Gnuastro’s own Table (see Section 5.3.1 [Column arithmetic], page 230). Therefore when typing (recommended)
1.12 Acknowledgments

Gnuastro would not have been possible without scholarships and grants from several funding institutions. We thus ask that if you used Gnuastro in any of your papers/reports, please add the proper citation and acknowledge the funding agencies/projects. For details of which papers to cite (may be different for different programs) and get the acknowledgment statement to include in your paper, please run the relevant programs with the common \texttt{--cite} option like the example commands below (for more on \texttt{--cite}, please see Section 4.1.2.3 [Operating mode options], page 162).

\begin{verbatim}
$ astnoisechisel --cite
$ astmkcatalog --cite
\end{verbatim}

Here, we’ll acknowledge all the institutions (and their grants) along with the people who helped make Gnuastro possible. The full list of Gnuastro authors is available at the start of this book and the \texttt{AUTHORS} file in the source code (both are generated automatically from the version controlled history). The plain text file \texttt{THANKS}, which is also distributed along with the source code, contains the list of people and institutions who played an indirect role in Gnuastro (not committed any code in the Gnuastro version controlled history).

The Japanese Ministry of Education, Culture, Sports, Science, and Technology (MEXT) scholarship for Mohammad Akhlaghi’s Masters and PhD degree in Tohoku University Astronomical Institute had an instrumental role in the long term learning and planning that made the idea of Gnuastro possible. The very critical view points of Professor Takashi Ichikawa (Mohammad’s adviser) were also instrumental in the initial ideas and creation of Gnuastro. Afterwards, the European Research Council (ERC) advanced grant 339659-MUSICOS (Principal investigator: Roland Bacon) was vital in the growth and expansion of Gnuastro. Working with Roland at the Centre de Recherche Astrophysique de Lyon (CRAL), enabled a thorough re-write of the core functionality of all libraries and programs, turning Gnuastro into the large collection of generic programs and libraries it is today. At the Instituto de Astrofisica de Canarias (IAC, and in particular in collaboration with Johan Knapp and Ignacio Trujillo), Gnuastro matured and its user base significantly grew. Work on improving Gnuastro is now continuing primarily in the Centro de Estudios de Física del Cosmos de Aragón (CEFCA), located in Teruel, Spain.

In general, we would like to gratefully thank the following people for their useful and constructive comments and suggestions (in alphabetical order by family name): Valentina Abrilmelgarejo, Marjan Akbari, Carlos Allende Prieto, Hamed Altafi, Roland Bacon, Roberto Baena Gallé, Zahra Bagheri, Karl Berry, Leindert Boogaard, Nicolas Bouché, Stefan Briëns, Fernando Buitrago, Adrian Bunk, Rosa Calvi, Mark Calabretta Nushkia Chamba, Sergio Chueca Urzay, Tamara Civera Lorenzo, Benjamin Clement, Nima Dehdilani, Andrés Del Pino Molina, Antonio Diaz Diaz, Alexey Dokuchaev, Pierre-Alain Duc, Alessandro Ederoclite, Elham Eftekharhi, Paul Eggert, Sepideh Eskandarlou, Juan Antonio Fernández Ontiveros, Gaspar Galaz, Andrés García-Serra Romero, Zohre Ghaffari, Thérèse Godefroy, Giulia Golini, Martin Guerrero Roncel, Madusha Gunawardhana, Bruno Haible, Stephen Hamer, Zahra Hosseini, Leslie Hunt, Takashi Ichikawa, Raúl Infante Sainz, Brandon Invergo, Oryna Ivashtenko, Aurélien Jarno, Lee Kelvin, Brandon Kelly, Mohammad-Reza Khellat,
Chapter 1: Introduction

Johan Knapen, Geoffry Krouchi, Martin Kuemmel, Clotilde Laigle, Floriane Leclercq, Alan Lefor, Javier Licandro, Alejandro Lumbresas Calle, Sebastián Luna Valero, Alberto Madrigal, Guillaume Mahler, Juan Miro, Alireza Molaeinezhad, Javier Moldon, Juan Molina Tobar, Francesco Montanari, Raphael Morales, Carlos Morales Socorro, Sylvain Mottet, Dmitrii Oparin, François Ochsenbein, Bertrand Pain, William Pence, Mamta Pommier, Marcel Popescu, Bob Proulx, Joseph Putko, Samane Raji, Ignacio Ruiz Cejudo, Teymoor Saifollahi, Joanna Sakowska, Elham Saremi, Markus Schaney, Yahya Sefidbakht, Alejandro Serrano Borlaff, Zahra Sharbaf, David Shupe, Leigh Smith, Jenny Sorce, Manuel Sánchez-Benavente, Lee Spitler, Richard Stallman, Michael Stein, Ole Streicher, Alfred M. Szmidt, Michel Tallon, Juan C. Tello, Vincenzo Testa, Éric Thiébaut, Ignacio Trujillo, Peter Teuben, David Valls-Gabaud, Jesús Varela, Aaron Watkins, Richard Wilbur, Michael H.F. Wilkinson, Christopher Willmer, Xiuqin Wu, Sara Yousefi Taemeh, Johannes Zabl. The GNU French Translation Team is also managing the French version of the top Gnuastro web page which we highly appreciate. Finally, we should thank all the (sometimes anonymous) people in various online forums who patiently answered all our small (but important) technical questions.

All work on Gnuastro has been voluntary, but the authors are most grateful to the following institutions (in chronological order) for hosting/supporting us in our research. Where necessary, these institutions have disclaimed any ownership of the parts of Gnuastro that were developed there, thus insuring the freedom of Gnuastro for the future (see Section 12.12.1 [Copyright assignment], page 693). We highly appreciate their support for free software, and thus free science, and therefore a free society.

Tohoku University Astronomical Institute, Sendai, Japan.
University of Salento, Lecce, Italy.
Centre de Recherche Astrophysique de Lyon (CRAL), Lyon, France.
Instituto de Astrofísica de Canarias (IAC), Tenerife, Spain.
Centro de Estudios de Física del Cosmos de Aragón (CEFCA), Teruel, Spain.
Google Summer of Code 2020 and 2021
2 Tutorials

To help new users have a smooth and easy start with Gnuastro, in this chapter several thoroughly elaborated tutorials, or cookbooks, are provided. These tutorials demonstrate the capabilities of different Gnuastro programs and libraries, along with tips and guidelines for the best practices of using them in various realistic situations.

We strongly recommend going through these tutorials to get a good feeling of how the programs are related (built in a modular design to be used together in a pipeline), very similar to the core Unix-based programs that they were modeled on. Therefore these tutorials will greatly help in optimally using Gnuastro’s programs (and generally, the Unix-like command-line environment) effectively for your research.

In Section 2.1 [Sufi simulates a detection], page 18, we’ll start with a fictional\(^1\) tutorial explaining how Abd al-rahman Sufi (903 – 986 A.D., the first recorded description of “nebulous” objects in the heavens is attributed to him) could have used some of Gnuastro’s programs for a realistic simulation of his observations and see if his detection of nebulous objects was trust-able. Because all conditions are under control in a simulated/mock environment/dataset, mock datasets can be a valuable tool to inspect the limitations of your data analysis and processing. But they need to be as realistic as possible, so the first tutorial is dedicated to this important step of an analysis.

The next two tutorials (Section 2.2 [General program usage tutorial], page 29, and Section 2.3 [Detecting large extended targets], page 76) use real input datasets from some of the deep Hubble Space Telescope (HST) images and the Sloan Digital Sky Survey (SDSS) respectively. Their aim is to demonstrate some real-world problems that many astronomers often face and how they can be solved with Gnuastro’s programs.

The ultimate aim of Section 2.2 [General program usage tutorial], page 29, is to detect galaxies in a deep HST image, measure their positions and brightness and select those with the strongest colors. In the process, it takes many detours to introduce you to the useful capabilities of many of the programs. So please be patient in reading it. If you don’t have much time and can only try one of the tutorials, we recommend this one.

Section 2.3 [Detecting large extended targets], page 76, deals with a major problem in astronomy: effectively detecting the faint outer wings of bright (and large) nearby galaxies to extremely low surface brightness levels (roughly one quarter of the local noise level in the example discussed). Besides the interesting scientific questions in these low-surface brightness features, failure to properly detect them will bias the measurements of the background objects and the survey’s noise estimates. This is an important issue, especially in wide surveys. Because bright/large galaxies and stars\(^2\), cover a significant fraction of the survey area.

\(^1\) The two historically motivated tutorials (Section 2.1 [Sufi simulates a detection], page 18, is not intended to be a historical reference (the historical facts of this fictional tutorial used Wikipedia as a reference). This form of presenting a tutorial was influenced by the PGF/TikZ and Beamer manuals. They are both packages in \TeX, the first is a high-level vector graphic programming environment, while with the second you can make presentation slides. On a similar topic, there are also some nice words of wisdom for Unix-like systems called Rootless Root (http://catb.org/esr/writings/unix-koans). These also have a similar style but they use a mythical figure named Master Foo. If you already have some experience in Unix-like systems, you will definitely find these Unix Koans entertaining/educative.

\(^2\) Stars also have similarly large and extended wings due to the point spread function, see Section 8.1.1.2 [Point spread function], page 442.
Section 2.4 [Building the extended PSF], page 97, tackles an important problem in astronomy: how to extract the PSF of an image, to the largest possible extent, without assuming any functional form. In Gnuastro we have multiple installed scripts for this job. Their usage and logic behind best tuning them for the particular step, is fully described in this tutorial, on a real dataset. The tutorial concludes with subtracting that extended PSF from the science image; thus giving you a cleaner image (with no scattered light of the brighter stars) for your higher-level analysis.

In these tutorials, we have intentionally avoided too many cross references to make it more easy to read. For more information about a particular program, you can visit the section with the same name as the program in this book. Each program section in the subsequent chapters starts by explaining the general concepts behind what it does, for example see Section 6.3 [Convolve], page 306. If you only want practical information on running a program, for example its options/configuration, input(s) and output(s), please consult the subsection titled “Invoking ProgramName”, for example see Section 7.2.2 [Invoking NoiseChisel], page 365. For an explanation of the conventions we use in the example codes through the book, please see Section 1.11 [Conventions], page 14.

2.1 Sufi simulates a detection

It is the year 953 A.D. and Abd al-rahman Sufi (903 – 986 A.D.)

3 In Latin Sufi is known as Azophi. He was an Iranian astronomer. His manuscript “Book of fixed stars” contains the first recorded observations of the Andromeda galaxy, the Large Magellanic Cloud and seven other non-stellar or ‘nebulous’ objects.

is in Shiraz as a guest astronomer. He had come there to use the advanced 123 centimeter astrolabe for his studies on the Ecliptic. However, something was bothering him for a long time. While mapping the constellations, there were several non-stellar objects that he had detected in the sky, one of them was in the Andromeda constellation. During a trip he had to Yemen, Sufi had seen another such object in the southern skies looking over the Indian ocean. He wasn’t sure if such cloud-like non-stellar objects (which he was the first to call ‘Sahābi’ in Arabic or ‘nebulous’) were real astronomical objects or if they were only the result of some bias in his observations. Could such diffuse objects actually be detected at all with his detection technique?

He still had a few hours left until nightfall (when he would continue his studies on the ecliptic) so he decided to find an answer to this question. He had thoroughly studied Claudius Ptolemy’s (90 – 168 A.D.) Almagest and had made lots of corrections to it, in particular in measuring the brightness. Using his same experience, he was able to measure a magnitude for the objects and wanted to simulate his observation to see if a simulated object with the same brightness and size could be detected in a simulated noise with the same detection technique. The general outline of the steps he wants to take are:

1. Make some mock profiles in an over-sampled image. The initial mock image has to be over-sampled prior to convolution or other forms of transformation in the image. Through his experiences, Sufi knew that this is because the image of heavenly bodies is actually transformed by the atmosphere or other sources outside the atmosphere (for example gravitational lenses) prior to being sampled on an image. Since that transformation occurs on a continuous grid, to best approximate it, he should do all
the work on a finer pixel grid. In the end he can re-sample the result to the initially
desired grid size.

2. Convolve the image with a point spread function (PSF, see Section 8.1.1.2 [Point spread
function], page 442) that is over-sampled to the same resolution as the mock image.
Since he wants to finish in a reasonable time and the PSF kernel will be very large due
to oversampling, he has to use frequency domain convolution which has the side effect
of dimming the edges of the image. So in the first step above he also has to build the
image to be larger by at least half the width of the PSF convolution kernel on each
edge.

3. With all the transformations complete, the image should be re-sampled to the same
size of the pixels in his detector.

4. He should remove those extra pixels on all edges to remove frequency domain convolu-
tion artifacts in the final product.

5. He should add noise to the (until now, noise-less) mock image. After all, all observations
have noise associated with them.

Fortunately Sufi had heard of GNU Astronomy Utilities from a colleague in Isfahan
(where he worked) and had installed it on his computer a year before. It had tools to do all
the steps above. He had used MakeProfiles before, but wasn’t sure which columns he had
chosen in his user or system wide configuration files for which parameters, see Section 4.2
[Configuration files], page 169. So to start his simulation, Sufi runs MakeProfiles with the
-P option to make sure what columns in a catalog MakeProfiles currently recognizes and
the output image parameters. In particular, Sufi is interested in the recognized columns
(shown below).

```
$ astmkprof -P
```

```
[[[ ... Truncated lines ... ]]]

# Output:
  type  float32  # Type of output: e.g., int16, float32, etc...
margedsize  1000,1000  # Number of pixels along first FITS axis.
oversample  5  # Scale of oversampling (>0 and odd).
[[[ ... Truncated lines ... ]]]

# Columns, by info (see `--searchin`), or number (starting from 1):
  ccol  2  # Coord. columns (one call for each dim.).
  ccol  3  # Coord. columns (one call for each dim.).
  fcol  4  # sersic (1), moffat (2), gaussian (3), point
    # (4), flat (5), circumference (6), distance
    # (7), custom-prof (8), azimuth (9),
    # custom-img (10).
  rcol  5  # Effective radius or FWHM in pixels.
  ncol  6  # Sersic index or Moffat beta.
  pcol  7  # Position angle.
  qcol  8  # Axis ratio.
```
In Gnuastro, column counting starts from 1, so the columns are ordered such that the first column (number 1) can be an ID he specifies for each object (and MakeProfiles ignores), each subsequent column is used for another property of the profile. It is also possible to use column names for the values of these options and change these defaults, but Sufi preferred to stick to the defaults. Fortunately MakeProfiles has the capability to also make the PSF which is to be used on the mock image and using the --prepforconv option, he can also make the mock image to be larger by the correct amount and all the sources to be shifted by the correct amount.

For his initial check he decides to simulate the nebula in the Andromeda constellation. The night he was observing, the PSF had roughly a FWHM of about 5 pixels, so as the first row (profile) in the table below, he defines the PSF parameters. Sufi sets the radius column (rcol above, fifth column) to 5.000, he also chooses a Moffat function for its functional form. Remembering how diffuse the nebula in the Andromeda constellation was, he decides to simulate it with a mock Sérsic index 1.0 profile. He wants the output to be 499 pixels by 499 pixels, so he can put the center of the mock profile in the central pixel of the image which is the 250th pixel along both dimensions (note that an even number doesn’t have a “central” pixel).

Looking at his drawings of it, he decides a reasonable effective radius for it would be 40 pixels on this image pixel scale (second row, 5th column below). He also sets the axis ratio (0.4) and position angle (-25 degrees) to approximately correct values too, and finally he sets the total magnitude of the profile to 3.44 which he had measured. Sufi also decides to truncate both the mock profile and PSF at 5 times the respective radius parameters. In the end he decides to put four stars on the four corners of the image at very low magnitudes as a visual scale. While he was preparing the catalog, one of his students approached him and was also following the steps.

As described above, the catalog of profiles to build will be a table (multiple columns of numbers) like below:

```
<p>| | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.000</td>
<td>0.000</td>
<td>2</td>
<td>5</td>
<td>4.7</td>
<td>0.0</td>
</tr>
<tr>
<td>1</td>
<td>250.0</td>
<td>250.0</td>
<td>1</td>
<td>40</td>
<td>1.0</td>
<td>-25</td>
</tr>
<tr>
<td>2</td>
<td>50.00</td>
<td>50.00</td>
<td>4</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>3</td>
<td>450.0</td>
<td>50.00</td>
<td>4</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>4</td>
<td>50.00</td>
<td>450.0</td>
<td>4</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>5</td>
<td>450.0</td>
<td>450.0</td>
<td>4</td>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
</tr>
</tbody>
</table>
```

This contains all the “data” to build the profile, and you can easily pass it to Gnuastro’s MakeProfiles: since Sufi already knows the columns and expected values very good, he has placed the information in the proper columns. However, when the student sees this, he just sees a mumble-jumble of numbers! Generally, Sufi explains to the student that even if you know the number positions and values very nicely today, in a couple of months you will forget! It will then be very hard for you to interpret the numbers properly. So you should never use naked data (or data without any extra information).
Data (or information) that describes other data is called “metadata”! One common example is column names (the name of a column is itself a data element, but data that describes the lower-level data within that column: how to interpret the numbers within it). Sufi explains to his student that Gnuastro has a convention for adding metadata within a plain-text file; and guides him to Section 4.7.2 [Gnuastro text table format], page 185.

Because we don’t want metadata to be confused with the actual data, in a plain-text file, we start lines containing metadata with a ‘#’. For example, see the same data above, but this time with metadata for every column:

```
# Column 1: ID [counter, u8] Identifier
# Column 2: X [pix, f32] Horizontal position
# Column 3: Y [pix, f32] Vertical position
# Column 4: PROFILE [name, u8] Radial profile function
# Column 5: R [pix, f32] Effective radius
# Column 6: N [n/a, f32] Sersic index
# Column 7: PA [deg, f32] Position angle
# Column 8: Q [n/a, f32] Axis ratio
# Column 9: MAG [log, f32] Magnitude
# Column 10: TRUNC [n/a, f32] Truncation (multiple of R)
```

```
0 0.000 0.000 2 5 4.7 0.0 1.0 30.0 5.0
1 250.0 250.0 1 40 1.0 -25 0.4 3.44 5.0
2 50.00 50.00 4 0 0.0 0.0 0.0 6.00 0.0
3 450.0 50.00 4 0 0.0 0.0 0.0 6.50 0.0
4 50.00 450.0 4 0 0.0 0.0 0.0 7.00 0.0
5 450.0 450.0 4 0 0.0 0.0 0.0 7.50 0.0
```

The numbers now make much more sense to for the student! Before continuing, Sufi reminded the student that even though metadata may not be strictly/technically necessary (for the computer programs), metadata are critical for human readers! Therefore, a good scientist should never forget to keep metadata with any data that they create, use or archive.

To start simulating the nebula, Sufi creates a directory named `simulationtest` in his home directory. Note that the `pwd` command will print the “parent working directory” of the current directory (its a good way to confirm/check your current location in the full file system: it always starts from the root, or `/`).

```
$ mkdir ~/simulationtest
$ cd ~/simulationtest
$ pwd
/home/rahman/simulationtest
```

It is possible to use a plain-text editor to manually put the catalog contents above into a plain-text file. But to easily automate catalog production (in later trials), Sufi decides to fill the input catalog with the redirection features of the command-line (or shell). Sufi’s student wasn’t familiar with this feature of the shell! So Sufi decided to do a fast demo; giving the following explanations while running the commands:

```
Shell redirection allows you to “re-direct” the “standard output” of a program (which is usually printed by the program on the command-line during its execution; like the output of `pwd` above) into a file. For example, let’s simply “echo” (or print to standard output) the line “This is a test.”:
```

```
$ echo "This is a test."
This is a test.

As you see, our statement was simply “echo”-ed to the standard output! To redirect this sentence into a file (instead of simply printing it on the standard output), we can simply use the > character, followed by the name of the file we want it to be dumped in.

$ echo "This is a test." > test.txt

This time, the `echo` command didn’t print anything in the terminal. Instead, the shell (command-line environment) took the output, and “re-directed” it into a file called `test.txt`. Let’s confirm this with the `ls` command (`ls` is short for “list” and will list all the files in the current directory):

$ ls
  test.txt

Now that you confirm the existence of `test.txt`, you can see its contents with the `cat` command (short for “concatenation”; because it can also merge multiple files together):

$ cat test.txt
This is a test.

Now that we have written our first line in `test.txt`, let’s try adding a second line (don’t forget that our final catalog of objects to simulate will have multiple lines):  

$ echo "This is my second line." > test.txt
$ cat test.txt
This is my second line.

As you see, the first line that you put in the file is no longer present! This happens because ‘>` always starts dumping content to a file from the start of the file. In effect, this means that any possibly pre-existing content is over-written by the new content! To append new lines (or dumping new content at the end of existing content), you can use ‘`>>`’. For example with the commands below, first we’ll write the first sentence (using ‘>`), then use ‘`>>`’ to add the second and third sentences. Finally, we’ll print the contents of `test.txt` to confirm that all three lines are preserved.

$ echo "My first sentence." > test.txt
$ echo "My second sentence." >> test.txt
$ echo "My third sentence." >> test.txt
$ cat test.txt
My first sentence.
My second sentence.
My third sentence.

The student thanked Sufi for this explanation and now feels more comfortable with redirection. Therefore Sufi continues with the main project. But before that, he deletes the temporary test file:

$ rm test.txt

To put the catalog of profile data and their metadata (that was described above) into a file, Sufi uses the commands below. While Sufi was writing these commands, the student complained that “I could have done in this in a text editor”. Sufi reminded the student that it is indeed possible; but it requires manual intervention. The advantage of a solution
like below is that it can be automated (for example adding more rows; for more profiles in the final image).

```bash
$ echo "# Column 1: ID [counter, u8] Identifier" > cat.txt
$ echo "# Column 2: X [pix, f32] Horizontal position" >> cat.txt
$ echo "# Column 3: Y [pix, f32] Vertical position" >> cat.txt
$ echo "# Column 4: PROF [name, u8] Radial profile function" \ >> cat.txt
$ echo "# Column 5: R [pix, f32] Effective radius" >> cat.txt
$ echo "# Column 6: N [n/a, f32] Sersic index" >> cat.txt
$ echo "# Column 7: PA [deg, f32] Position angle" >> cat.txt
$ echo "# Column 8: Q [n/a, f32] Axis ratio" >> cat.txt
$ echo "# Column 9: MAG [log, f32] Magnitude" >> cat.txt
$ echo "# Column 10: TRUNC [n/a, f32] Truncation (multiple of R)" \ >> cat.txt
$ echo "0 0.000 0.000 2 5 4.7 0.0 1.0 30.0 5.0" >> cat.txt
$ echo "1 250.0 250.0 1 40 1.0 -25 0.4 3.44 5.0" >> cat.txt
$ echo "2 50.00 50.00 4 0 0.0 0.0 0.0 6.00 0.0" >> cat.txt
$ echo "3 450.0 50.00 4 0 0.0 0.0 0.0 6.50 0.0" >> cat.txt
$ echo "4 50.00 450.0 4 0 0.0 0.0 0.0 7.00 0.0" >> cat.txt
$ echo "5 450.0 450.0 4 0 0.0 0.0 0.0 7.50 0.0" >> cat.txt
```

To make sure if the catalog’s content is correct (and there was no typo for example!), Sufi runs `cat cat.txt`, and confirms that it is correct.

Now that the catalog is created, Sufi is ready to call MakeProfiles to build the image containing these objects. He looks into his records and finds that the zero point magnitude for that night and that detector 18. The student was a little confused on the concept of zero point, so Sufi pointed him to Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394, which the student can study in detail later. Sufi therefore runs MakeProfiles with the command below:

```bash
$ astmkprof --prepforconv --mergedsize=499,499 --zeropoint=18.0 cat.txt
MakeProfiles 0.17.44-ad2d9 started on Sat Oct 6 16:26:56 953
- 6 profiles read from cat.txt
- Random number generator (RNG) type: ranlxs1
- Basic RNG seed: 1652884540
- Using 12 threads.
- row 3 complete, 5 left to go
- row 4 complete, 4 left to go
- row 6 complete, 3 left to go
- row 5 complete, 2 left to go
- ./0_cat_profiles.fits created.
- row 1 complete, 1 left to go
- row 2 complete, 0 left to go
- ./cat_profiles.fits created. 0.092573 seconds
```

```
--- Output: ./cat_profiles.fits
MakeProfiles finished in 0.293644 seconds
```
Sufi encourages the student to read through the printed output. As the statements say, two FITS files should have been created in the running directory. So Sufi ran the command below to confirm:

```
$ ls
0_cat_profiles.fits  cat_profiles.fits  cat.txt
```

The file `0_cat_profiles.fits` is the PSF Sufi had asked for, and `cat_profiles.fits` is the image containing the main objects in the catalog. Sufi opened the main image with the command below (using SAO DS9):

```
$ astscript-fits-view cat_profiles.fits --ds9scale=95
```

The student could clearly see the main elliptical structure in the center. However, the size of `cat_profiles.fits` was surprising for the student, instead of 499 by 499 (as we had requested), it was 2615 by 2615 pixels (from the command below):

```
$ astfits cat_profiles.fits
```

Fits (GNU Astronomy Utilities) 0.17.44-ad2d9
Run on Sat Oct  6 16:26:58 953

HDU (extension) information: 'cat_profiles.fits'.
Column 1: Index (counting from 0, usable with `--hdu').
Column 2: Name (`EXTNAME' in FITS standard, usable with `--hdu').
Column 3: Image data type or 'table' format (ASCII or binary).
Column 4: Size of data in HDU.

```
0  MKPROF-CONFIG  no-data  0
1  Mock profiles  float32  2615x2615
```

So Sufi explained why oversampling is important in modeling, especially for parts of the image where the flux change is significant over a pixel. Recall that when you oversample the model (for example by 5 times), for every desired pixel, you get 25 pixels \((5 \times 5)\). Sufi then explained that after convolving (next step below) we will down-sample the image to get our originally desired size/resolution.

After seeing the image, the student complained that only the large elliptical model for the Andromeda nebula can be seen in the center. He couldn’t see the four stars that we had also requested in the catalog. So Sufi had to explain that the stars are there in the image, but the reason that they aren’t visible when looking at the whole image at once, is that they only cover a single pixel! To prove it, he centered the image around the coordinates 2308 and 2308, where one of the stars is located in the over-sampled image [you can do this in `ds9` by selecting “Pan” in the “Edit” menu, then clicking around that position]. Sufi then zoomed in to that region and soon, the star’s non-zero pixel could be clearly seen.

Sufi explained that the stars will take the shape of the PSF (cover an area of more than one pixel) after convolution. If we didn’t have an atmosphere and we didn’t need an aperture, then stars would only cover a single pixel with normal CCD resolutions. So Sufi convolved the image with this command:

```
$ astconvolve --kernel=0_cat_profiles.fits cat_profiles.fits \
  --output=cat_convolved.fits
```

Convolve started on Sat Oct  6 16:35:32 953
- Using 8 CPU threads.
- Input: cat_profiles.fits (hdu: 1)
- Kernel: 0_cat_profiles.fits (hdu: 1)
- Input and Kernel images padded. 0.075541 seconds
- Images converted to frequency domain. 6.728407 seconds
- Multiplied in the frequency domain. 0.040659 seconds
- Converted back to the spatial domain. 3.465344 seconds
- Padded parts removed. 0.016767 seconds
- Output: cat_convolved.fits

Convolve finished in: 10.422161 seconds

When convolution finished, Sufi opened cat_convolved.fits and the four stars could be easily seen now:

```
$ astscript-fits-view cat_convolved.fits --ds9scale=95
```

It was interesting for the student that all the flux in that single pixel is now distributed over so many pixels (the sum of all the pixels in each convolved star is actually equal to the value of the single pixel before convolution). Sufi explained how a PSF with a larger FWHM would make the points even wider than this (distributing their flux in a larger area). With the convolved image ready, they were prepared to re-sample it to the original pixel scale Sufi had planned [from the `$ astmkprof -P` command above, recall that MakeProfiles had over-sampled the image by 5 times]. Sufi explained the basic concepts of warping the image to his student and ran Warp with the following command:

```
$ astwarp --scale=1/5 --centeroncorner cat_convolved.fits
```

Warp started on Sat Oct 6 16:51:59 953
Using 8 CPU threads.
Input: cat_convolved.fits (hdu: 1)
matrix:
```
  0.2000  0.0000  0.4000
  0.0000  0.2000  0.4000
  0.0000  0.0000  1.0000
```

```
$ astfits cat_convolved_scaled.fits --quiet
0    WARP-CONFIG no-data 0
1    Warped float32 523x523
```

cat_convolved_scaled.fits now has the correct pixel scale. However, the image is still larger than what we had wanted, it is 523 × 523 pixels (not our desired 499 × 499). The student is slightly confused, so Sufi also re-samples the PSF with the same scale by running

```
$ astwarp --scale=1/5 --centeroncorner 0_cat_profiles.fits
$ astfits 0_cat_profiles_scaled.fits --quiet
```

```
0    WARP-CONFIG no-data 0
1    Warped float32 25x25
```

Sufi notes that 25 = 12 + 12 + 1 and that 523 = 499 + 12 + 12. He goes on to explain that frequency space convolution will dim the edges and that is why he added the `--prepforconv` option to MakeProfiles above. Now that convolution is done, Sufi can remove those extra pixels using Crop with the command below. Crop’s `--section` option accepts coordinates inclusively and counting from 1 (according to the FITS standard), so the crop region’s first pixel has to be 13, not 12.
$ astcrop cat_convolved_scaled.fits --section=13:*-12,13:*-12 \\
--mode=img --zeroisnotblank
Crop started on Sat Oct 6 17:03:24 953
- Read metadata of 1 image. 0.001304 seconds
---- ...nvolved_scaled_cropped.fits created: 1 input.
Crop finished in: 0.027204 seconds

To fully convince the student, Sufi checks the size of the output of the crop command above:

$ astfits cat_convolved_scaled_cropped.fits --quiet
0  n/a  no-data  0
1  n/a  float32  499x499

Finally, `cat_convolved_scaled_cropped.fits` is 499 × 499 pixels and the mock Andromeda galaxy is centered on the central pixel. This is the same dimensions as Sufi had desired in the beginning. All this trouble was certainly worth it because now there is no dimming on the edges of the image and the profile centers are more accurately sampled.

The final step to simulate a real observation would be to add noise to the image. Sufi set the zero point magnitude to the same value that he set when making the mock profiles and looking again at his observation log, he had measured the background flux near the nebula had a magnitude of 7 that night. For more on how the background value determines the noise, see Section 8.2.1 [Noise basics], page 461. So using these values he ran MakeNoise, and with the second command, he visually inspected the image.

$ astmknoise --zeropoint=18 --background=7 --output=out.fits \\
cat_convolved_scaled_cropped.fits
MakeNoise 0.17.44-ad2d9 started on Sat Oct 6 17:05:06 953
- Generator type: ranlxs1
- Generator seed: 1428318100
MakeNoise finished in: 0.033491 (seconds)

$ astscript-fits-view out.fits
The `out.fits` file now contains the noised image of the mock catalog Sufi had asked for. The student hadn’t observed the nebula in the sky, so when he saw the mock image in SAO DS9 (with the second command above), he understood why Sufi was dubious: it was very diffuse!

Seeing how the `--output` option allows the user to specify the name of the output file, the student was confused and wanted to know why Sufi hadn’t used it more regularly before? Sufi explained that for intermediate steps, you can rely on the automatic output of the programs (see Section 4.9 [Automatic output], page 191). Doing so will give all the intermediate files a similar basic name structure, so in the end you can simply remove them all with the Shell’s capabilities, and it will be familiar for other users. So Sufi decided to show this to the student by making a shell script from the commands he had used before.

The command-line shell has the capability to read all the separate input commands from a file. This is useful when you want to do the same thing multiple times, with only the names of the files or minor parameters changing between the different instances. Using the shell’s history (by pressing the up keyboard key) Sufi reviewed all the commands and then he retrieved the last 5 commands with the `history 5` command. He selected all those lines he had input and put them in a text file named `mymock.sh`. Then he defined the `edge`
and `base` shell variables for easier customization later. Finally, before every command, he added some comments (lines starting with `#`) for future readability.

```bash
edge=12
base=cat

# Stop running next commands if one fails.
set -e

# Remove any (possibly) existing output (from previous runs)
# before starting.
rm -f out.fits

# Run MakeProfiles to create an oversampled FITS image.
astmkprof --prepforconv --mergedsize=499,499 --zeropoint=18.0 "$base”.txt

# Convolve the created image with the kernel.
astconvolve "$base".profiles.fits \ 
   --kernel=0_"$base".profiles.fits \ 
   --output="$base".convolved.fits

# Scale the image back to the intended resolution.
astwarp --scale=1/5 --centeroncorner "$base".convolved.fits

# Crop the edges out (dimmed during convolution). ' --section' # accepts inclusive coordinates, so the start of the section # must be one pixel larger than its end.
st_edge=$(( edge + 1 ))
astcrop "$base".convolved_scaled.fits --zeroisnotblank \ 
   --mode=img --section=$st_edge:*-$edge,$st_edge:*-$edge

# Add noise to the image.
astmknoise --zeropoint=18 --background=7 --output=out.fits \ 
   "$base".convolved_scaled_cropped.fits

# Remove all the temporary files.
rm 0*.fits "$base"*.fits
```

He used this chance to remind the student of the importance of comments in code or shell scripts! Just like metadata in a dataset, when writing the code, you have a good mental picture of what you are doing, so writing comments might seem superfluous and excessive. However, in one month when you want to re-use the script, you have lost that mental picture and remembering it can be time-consuming and frustrating. The importance of comments is further amplified when you want to share the script with a friend/colleague. So it is good to accompany any step of a script, or code, with useful comments while you are writing it (create a good mental picture of why you are doing something: don’t just describe the command, but its purpose).
Sufi then explained to the eager student that you define a variable by giving it a name, followed by an = sign and the value you want. Then you can reference that variable from anywhere in the script by calling its name with a $ prefix. So in the script whenever you see $base, the value we defined for it above is used. If you use advanced editors like GNU Emacs or even simpler ones like Gedit (part of the GNOME graphical user interface) the variables will become a different color which can really help in understanding the script. We have put all the $base variables in double quotation marks ("”) so the variable name and the following text do not get mixed, the shell is going to ignore the " after replacing the variable value. To make the script executable, Sufi ran the following command:

$ chmod +x mymock.sh

Then finally, Sufi ran the script, simply by calling its file name:

$ ./mymock.sh

After the script finished, the only file remaining is the out.fits file that Sufi had wanted in the beginning. Sufi then explained to the student how he could run this script anywhere that he has a catalog if the script is in the same directory. The only thing the student had to modify in the script was the name of the catalog (the value of the base variable in the start of the script) and the value to the edge variable if he changed the PSF size. The student was also happy to hear that he won’t need to make it executable again when he makes changes later, it will remain executable unless he explicitly changes the executable flag with chmod.

The student was really excited, since now, through simple shell scripting, he could really speed up his work and run any command in any fashion he likes allowing him to be much more creative in his works. Until now he was using the graphical user interface which doesn’t have such a facility and doing repetitive things on it was really frustrating and some times he would make mistakes. So he left to go and try scripting on his own computer. He later reminded Sufi that the second tutorial in the Gnuastro book as more complex commands in data analysis, and a more advanced introduction to scripting (see Section 2.2 [General program usage tutorial], page 29).

Sufi could now get back to his own work and see if the simulated nebula which resembled the one in the Andromeda constellation could be detected or not. Although it was extremely faint4. Therefore, Sufi ran Gnuastro’s detection software (Section 7.2 [NoiseChisel], page 362) to see if this object is detectable or not. NoiseChisel’s output (out_detected.fits) is a multi-extension FITS file, so he used Gnuastro’s astscript-fits-view program in the second command to see the output:

$ astnoisechisel out.fits

$ astscript-fits-view out_detected.fits

In the “Cube” window (that was opened with DS9), if Sufi clicked on the “Next” button to see the pixels that were detected to contain significant signal. Fortuantely the nebula’s shape was detectable and he coud finally confirm that the nebula he kept in his notebook

---

4 The brightness of a diffuse object is added over all its pixels to give its final magnitude, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394. So although the magnitude 3.44 (of the mock nebula) is orders of magnitude brighter than 6 (of the stars), the central galaxy is much fainter. Put another way, the brightness is distributed over a large area in the case of a nebula.
was actually observable. He wrote this result in the draft manuscript that would later become “Book of fixed stars”\(^5\).

He still had to check the other nebula he saw from Yemen and several other such objects, but they could wait until tomorrow (thanks to the shell script, he only has to define a new catalog). It was nearly sunset and they had to begin preparing for the night’s measurements on the ecliptic.

### 2.2 General program usage tutorial

Measuring colors of astronomical objects in broad-band or narrow-band images is one of the most basic and common steps in astronomical analysis. Here, we will use Gnuastro’s programs to get a physical scale (area at certain redshifts) of the field we are studying, detect objects in a Hubble Space Telescope (HST) image, measure their colors and identify the ones with the strongest colors, do a visual inspection of these objects and inspect spatial position in the image. After this tutorial, you can also try the Section 2.3 [Detecting large extended targets], page 76, tutorial which goes into a little more detail on detecting very low surface brightness signal.

During the tutorial, we will take many detours to explain, and practically demonstrate, the many capabilities of Gnuastro’s programs. In the end you will see that the things you learned during this tutorial are much more generic than this particular problem and can be used in solving a wide variety of problems involving the analysis of data (images or tables). So please don’t rush, and go through the steps patiently to optimally master Gnuastro.

In this tutorial, we’ll use the HSTeXtreme Deep Field (https://archive.stsci.edu/prepds/xf) dataset. Like almost all astronomical surveys, this dataset is free for download and usable by the public. You will need the following tools in this tutorial: Gnuastro, SAO DS9\(^6\), GNU Wget\(^7\), and AWK (most common implementation is GNU AWK\(^8\)).

This tutorial was first prepared for the “Exploring the Ultra-Low Surface Brightness Universe” workshop (November 2017) at the ISSI in Bern, Switzerland. It was further extended in the “4th Indo-French Astronomy School” (July 2018) organized by LIO, CRAL CNRS UMR5574, UCBL, and IUCAA in Lyon, France. We are very grateful to the organizers of these workshops and the attendees for the very fruitful discussions and suggestions that made this tutorial possible.

---

**Write the example commands manually:** Try to type the example commands on your terminal manually and use the history feature of your command-line (by pressing the “up” button to retrieve previous commands). Don’t simply copy and paste the commands shown here. This will help simulate future situations when you are processing your own datasets.

---


\(^6\) See Section B.1 [SAO DS9], page 702, available at [http://ds9.si.edu/site/Home.html](http://ds9.si.edu/site/Home.html)

\(^7\) [https://www.gnu.org/software/wget](https://www.gnu.org/software/wget)

\(^8\) [https://www.gnu.org/software/gawk](https://www.gnu.org/software/gawk)
2.2.1 Calling Gnuastro’s programs

A handy feature of Gnuastro is that all program names start with ast. This will allow your command-line processor to easily list and auto-complete Gnuastro’s programs for you. Try typing the following command (press \texttt{TAB} key when you see \texttt{<TAB>}) to see the list:

\begin{verbatim}
$ ast<TAB><TAB>
\end{verbatim}

Any program that starts with ast (including all Gnuastro programs) will be shown. By choosing the subsequent characters of your desired program and pressing \texttt{<TAB><TAB>} again, the list will narrow down and the program name will auto-complete once your input characters are unambiguous. In short, you often don’t need to type the full name of the program you want to run.

2.2.2 Accessing documentation

Gnuastro contains a large number of programs and it is natural to forget the details of each program’s options or inputs and outputs. Therefore, before starting the analysis steps of this tutorial, let’s review how you can access this book to refresh your memory any time you want, without having to take your hands off the keyboard.

When you install Gnuastro, this book is also installed on your system along with all the programs and libraries, so you don’t need an internet connection to access/read it. Also, by accessing this book as described below, you can be sure that it corresponds to your installed version of Gnuastro.

GNU Info\footnote{GNU Info is already available on almost all Unix-like operating systems.} is the program in charge of displaying the manual on the command-line (for more, see Section 4.3.4 [Info], page 174). To see this whole book on your command-line, please run the following command and press subsequent keys. Info has its own mini-environment, therefore we’ll show the keys that must be pressed in the mini-environment after a \texttt{->} sign. You can also ignore anything after the \# sign in the middle of the line, they are only for your information.

\begin{verbatim}
$ info gnuastro # Open the top of the manual.
-> <SPACE> # All the book chapters.
-> <SPACE> # Continue down: show sections.
-> <SPACE> ... # Keep pressing space to go down.
-> q # Quit Info, return to the command-line.
\end{verbatim}

The thing that greatly simplifies navigation in Info is the links (regions with an underline). You can immediately go to the next link in the page with the \texttt{<TAB>} key and press \texttt{<ENTER>} on it to go into that part of the manual. Try the commands above again, but this time also use \texttt{<TAB>} to go to the links and press \texttt{<ENTER>} on them to go to the respective section of the book. Then follow a few more links and go deeper into the book. To return to the previous page, press 1 (small L). If you are searching for a specific phrase in the whole book (for example an option name), press \texttt{s} and type your search phrase and end it with an \texttt{<ENTER>}

You don’t need to start from the top of the manual every time. For example, to get to Section 7.2.2 [Invoking NoiseChisel], page 365, run the following command. In general, all programs have such an “Invoking ProgramName” section in this book. These sections are
specifically for the description of inputs, outputs and configuration options of each program. You can access them directly for each program by giving its executable name to Info.

$ info astnoisechisel

The other sections don’t have such shortcuts. To directly access them from the command-line, you need to tell Info to look into Gnuastro’s manual, then look for the specific section (an unambiguous title is necessary). For example, if you only want to review/remember NoiseChisels’s Section 7.2.2.2 [Detection options], page 370), just run the following command. Note how case is irrelevant for Info when calling a title in this manner.

$ info gnuastro "Detection options"

In general, Info is a powerful and convenient way to access this whole book with detailed information about the programs you are running. If you are not already familiar with it, please run the following command and just read along and do what it says to learn it. Do not stop until you feel sufficiently fluent in it. Please invest the half an hour’s time necessary to start using Info comfortably. It will greatly improve your productivity and you will start reaping the rewards of this investment very soon.

$ info info

As a good scientist you need to feel comfortable to play with the features/options and avoid (be critical to) using default values as much as possible. On the other hand, our human memory is limited, so it is important to be able to easily access any part of this book fast and remember the option names, what they do and their acceptable values.

If you just want the option names and a short description, calling the program with the --help option might also be a good solution like the first example below. If you know a few characters of the option name, you can feed the output to grep like the second or third example commands.

$ astnoisechisel --help
$ astnoisechisel --help | grep quant
$ astnoisechisel --help | grep check

### 2.2.3 Setup and data download

The first step in the analysis of the tutorial is to download the necessary input datasets. First, to keep things clean, let’s create a gnuastro-tutorial directory and continue all future steps in it:

$ mkdir gnuastro-tutorial
$ cd gnuastro-tutorial

We will be using the near infra-red Wide Field Camera (http://www.stsci.edu/hst/wfc3) dataset. If you already have them in another directory (for example XDFDIR, with the same FITS file names), you can set the download directory to be a symbolic link to XDFDIR with a command like this:

$ ln -s XDFDIR download

Otherwise, when the following images aren’t already present on your system, you can make a download directory and download them there.

$ mkdir download
$ cd download
$ xdfurl=http://archive.stsci.edu/pub/hlsp/xdf
In this tutorial, we’ll just use these three filters. Later, you may need to download more filters. To do that, you can use the shell’s for loop to download them all in series (one after the other) with one command like the one below for the WFC3 filters. Put this command instead of the three wget commands above. Recall that all the extra spaces, back-slashes (\), and new lines can be ignored if you are typing on the lines on the terminal.

```bash
$ for f in f105w f125w f140w f160w; do \
  wget $xdfurl/hlsp_xdf_hst_wfc3ir-60mas_hudf_"$f"_v1_sci.fits; \
done
```

### 2.2.4 Dataset inspection and cropping

First, let’s visually inspect the datasets we downloaded in Section 2.2.3 [Setup and data download], page 31. Let’s take F160W image as an example. Do the steps below with the other image(s) too (and later with any dataset that you want to work on). It is very important to get a good visual feeling of the dataset you intend to use. Also, note how SAO DS9 (used here for visual inspection of FITS images) doesn’t follow the GNU style of options where “long” and “short” options are preceded by -- and - respectively (for example --width and -w, see Section 4.1.1.2 [Options], page 154).

Run the command below to see the F160W image with DS9. Ds9’s -zscale scaling is good to visually highlight the low surface brightness regions, and as the name suggests, -zoom to fit will fit the whole dataset in the window. If the window is too small, expand it with your mouse, then press the “zoom” button on the top row of buttons above the image. Afterwards, in the bottom row of buttons, press “zoom fit”. You can also zoom in and out by scrolling your mouse or the respective operation on your touch-pad when your cursor/pointer is over the image.

```bash
$ ds9 download/hlsp_xdf_hst_wfc3ir-60mas_hudf_f160w_v1_sci.fits \
   -zscale -zoom to fit
```

As you hover your mouse over the image, notice how the “Value” and positional fields on the top of the ds9 window get updated. The first thing you might notice is that when you hover the mouse over the regions with no data, they have a value of zero. The next thing might be that the dataset actually has two “depth”s (see Section 7.4.3 [Quantifying measurement limits], page 398). Recall that this is a combined/reduced image of many exposures, and the parts that have more exposures are deeper. In particular, the exposure time of the deep inner region is larger than 4 times of the outer (more shallow) parts.

To simplify the analysis in this tutorial, we’ll only be working on the deep field, so let’s crop it out of the full dataset. Fortunately the XDF survey web page (above) contains the vertices of the deep flat WFC3-IR field. With Gnaastro’s Crop program\(^\text{11}\), you can use those vertices to cutout this deep region from the larger image. But before that, to keep

---

\(^{10}\) Note that you only have one port to the internet, so downloading in parallel will actually be slower than downloading in series.

\(^{11}\) To learn more about the crop program see Section 6.1 [Crop], page 259.
things organized, let’s make a directory called `flat-ir` and keep the flat (single-depth) regions in that directory (with a `xdf-` suffix for a shorter and easier filename).

```bash
$ mkdir flat-ir
$ astcrop --mode=wcs -h0 --output=flat-ir/xdf-f105w.fits \
   --polygon="53.187414,-27.779152 : 53.159507,-27.759633 : \n   53.134517,-27.787144 : 53.161906,-27.807208" \
   download/hlsp_xdf_hst_wfc3ir-60mas_hudf_f105w_v1_sci.fits

$ astcrop --mode=wcs -h0 --output=flat-ir/xdf-f125w.fits \
   --polygon="53.187414,-27.779152 : 53.159507,-27.759633 : \n   53.134517,-27.787144 : 53.161906,-27.807208" \
   download/hlsp_xdf_hst_wfc3ir-60mas_hudf_f125w_v1_sci.fits

$ astcrop --mode=wcs -h0 --output=flat-ir/xdf-f160w.fits \
   --polygon="53.187414,-27.779152 : 53.159507,-27.759633 : \n   53.134517,-27.787144 : 53.161906,-27.807208" \
   download/hlsp_xdf_hst_wfc3ir-60mas_hudf_f160w_v1_sci.fits
```

The only thing varying in the three calls to Gnuastro’s Crop program is the filter name! Note how everything else is the same. In such cases, you should generally avoid repeating a command manually, it is prone to many bugs, and as you see, it is very hard to read (didn’t you suddenly write a 7 as an 8?). To simplify the command, and later allow work on more filters, we can use the shell’s `for` loop as shown below. Notice how the place where the filter names (`f105w`, `f125w` and `f160w`) are used above, have been replaced with `$f` (the shell variable that `for` will update in every loop) below.

```bash
$ rm flat-ir/*.fits
$ for f in f105w f125w f160w; do \ 
   astcrop --mode=wcs -h0 --output=flat-ir/xdf-$f.fits \
   --polygon="53.187414,-27.779152 : 53.159507,-27.759633 : \n   53.134517,-27.787144 : 53.161906,-27.807208" \
   download/hlsp_xdf_hst_wfc3ir-60mas_hudf_$f_v1_sci.fits; \ 
done
```

Please open these images and inspect them with the same `ds9` command you used above. You will see how it is nicely flat now and doesn’t have varying depths. In the example above, the polygon vertices are in degrees, but you can also replace them with sexagesimal coordinates (for example using `03h32m44.9794` or `03:32:44.9794` instead of `53.187414` instead of the first RA and `-27d46m44.9472` or `-27:46:44.9472` instead of the first Dec). To further simplify things, you can even define your polygon visually as a DS9 “region”, save it as a “region file” and give that file to crop. But we need to continue, so if you are interested to learn more, see Section 6.1 [Crop], page 259.

Another important result of this crop is that regions with no data now have a NaN (Not-a-Number, or a blank value) value. In the downloaded files, such regions had a value of zero. However, zero is a number, and is thus meaningful, especially when you later want to NoiseChisel\(^\text{12}\). Generally, when you want to ignore some pixels in a dataset, and avoid

\(^{12}\) As you will see below, unlike most other detection algorithms, NoiseChisel detects the objects from their faintest parts, it doesn’t start with their high signal-to-noise ratio peaks. Since the Sky is already
higher-level ambiguities or complications, it is always best to give them blank values (not zero, or some other absurdly large or small number). Gnuastro has the Arithmetic program for such cases, and we’ll introduce it later in this tutorial.

### 2.2.5 Angular coverage on the sky

This is the deepest image we currently have of the sky. The first thing that comes to mind may be this: “How large is this field on the sky?” You can get a fast and crude answer with Gnuastro’s Fits program using this command:

```
astfits flat-ir/xdf-f160w.fits --skycoverage
```

It will print the sky coverage in two formats (all numbers are in units of degrees for this image): 1) the image’s central RA and Dec and full width around that center, 2) the range of RA and Dec covered by this image. You can use these values in various online query systems. You can also use this option to automatically calculate the area covered by this image. With the `--quiet` option, the printed output of `--skycoverage` will not contain human-readable text, making it easier for further processing:

```
astfits flat-ir/xdf-f160w.fits --skycoverage --quiet
```

The second row is the coverage range along RA and Dec (compare with the outputs before using `--quiet`). We can thus simply subtract the second from the first column and multiply it with the difference of the fourth and third columns to calculate the image area. We’ll also multiply each by 60 to have the area in arc-minutes squared.

```
astfits flat-ir/xdf-f160w.fits --skycoverage --quiet | awk 'NR==2{print ($2-$1)*60*($4-$3)*60}'
```

The returned value is 9.06711 arcmin². **However, this method ignores the fact many of the image pixels are blank!** In other words, the image does cover this area, but there is no data in more than half of the pixels. So let’s calculate the area coverage over-which we actually have data.

The FITS world coordinate system (WCS) meta data standard contains the key to answering this question. Run the following command to see all the FITS keywords (metadata) for one of the images (almost identical with the other images because they were are scaled to the same region of Sky):

```
astfits flat-ir/xdf-f160w.fits -h1
```

Look into the keywords grouped under the ‘World Coordinate System (WCS)’ title. These keywords define how the image relates to the outside world. In particular, the `CDELT*` subtracted in many images and noise fluctuates around zero, zero is commonly higher than the initial threshold applied. Therefore not ignoring zero-valued pixels in this image, will cause them to part of the detections!
keywords (or \texttt{CDELT1} and \texttt{CDELT2} in this 2D image) contain the “Coordinate DELTa” (or change in coordinate units) with a change in one pixel. But what is the units of each “world” coordinate? The \texttt{CUNIT*} keywords (for “Coordinate UNIT”) have the answer. In this case, both \texttt{CUNIT1} and \texttt{CUNIT2} have a value of \texttt{deg}, so both “world” coordinates are in units of degrees. We can thus conclude that the value of \texttt{CDELT*} is in units of degrees-per-pixel\footnote{With the FITS \texttt{CDELT} convention, rotation (PC or CD keywords) and scales (CDELT) are separated. In the FITS standard the CDELT keywords are optional. When CDELT keywords aren’t present, the PC matrix is assumed to contain both the coordinate rotation and scales. Note that not all FITS writers use the CDELT convention. So you might not find the CDELT keywords in the WCS meta data of some FITS files. However, all Gnuastro programs (which use the default FITS keyword writing format of WCSLIB) write their output WCS with the CDELT convention, even if the input doesn’t have it. If your dataset doesn’t use the CDELT convention, you can feed it to any (simple) Gnuastro program (for example Arithmetic) and the output will have the CDELT keyword. See Section 8 of the FITS standard (https://fits.gsfc.nasa.gov/standard40/fits_standard40aa-1e.pdf) for more information.}.

With the commands below, we’ll use \texttt{CDELT} (along with the image size) to find the answer of our initial question: “how much of the sky does this image cover?”. The lines starting with \texttt{##} are just comments for you to read and understand each command. Don’t type them on the terminal. The commands are intentionally repetitive in some places to better understand each step and also to demonstrate the beauty of command-line features like history, variables, pipes and loops (which you will commonly use as you master the command-line).

\begin{quote}
\textbf{Use shell history}: Don’t forget to make effective use of your shell’s history: you don’t have to re-type previous command to add something to them. This is especially convenient when you just want to make a small change to your previous command. Press the “up” key on your keyboard (possibly multiple times) to see your previous command(s) and modify them accordingly.
\end{quote}

\begin{verbatim}
## If your system language uses ',' (not '.') as decimal separator.
$ export LANG=C

## See the general statistics of non-blank pixel values.
$ aststatistics flat-ir/xf-f160w.fits

## We only want the number of non-blank pixels.
$ aststatistics flat-ir/xf-f160w.fits --number

## Keep the result of the command above in the shell variable ‘n’.
$ n=$(aststatistics flat-ir/xf-f160w.fits --number)

## See what is stored the shell variable ‘n’.
$ echo $n

## Show all the FITS keywords of this image.
$ astfits flat-ir/xf-f160w.fits -h1

## The resolution (in degrees/pixel) is in the ‘CDELT’ keywords.
\end{verbatim}
## Only show lines that contain these characters, by feeding
## the output of the previous command to the ‘grep’ program.
$ astfits flat-ir/xdf-f160w.fits -h1 | grep CDELT

## Since the resolution of both dimensions is (approximately) equal,
## we'll only use one of them (CDELT1).
$ astfits flat-ir/xdf-f160w.fits -h1 | grep CDELT1

## To extract the value (third token in the line above), we'll
## feed the output to AWK. Note that the first two tokens are
## ‘CDELT1’ and ‘=’.
$ astfits flat-ir/xdf-f160w.fits -h1 | grep CDELT1 | awk '{print $3}'

## Save it as the shell variable ‘r’.
$ r=$(astfits flat-ir/xdf-f160w.fits -h1 | grep CDELT1 \                        
   | awk '{print $3}')

## Print the values of ‘n’ and ‘r’.
$ echo $n $r

## Use the number of pixels (first number passed to AWK) and
## length of each pixel’s edge (second number passed to AWK)
## to estimate the area of the field in arc-minutes squared.
$ echo $n $r | awk '{print $1 * ($2*60)^2}'

The output of the last command (area of this field) is 4.03817 (or approximately 4.04)
arc-minutes squared. Just for comparison, this is roughly 175 times smaller than the average
moon’s angular area (with a diameter of 30arc-minutes or half a degree).

Some FITS writers don’t use the CDELT convention, making it hard to use the steps
above. In such cases, you can extract the pixel scale with the --pixelscale option of
Gnuastro’s Fits program like the command below. Like the --skycoverage option above,
you can also use the --quiet option to allow easy usage of the values in scripts.

$ astfits flat-ir/xdf-f160w.fits --pixelscale

### AWK for table/value processing:
As you saw above AWK is a powerful and simple tool
for text processing. You will see it often in shell scripts. GNU AWK (the most common
implementation) comes with a free and wonderful book ([https://www.gnu.org/software/gawk/manual/](https://www.gnu.org/software/gawk/manual/)) in the same format as this book which will allow you to master it nicely. Just
like this manual, you can also access GNU AWK’s manual on the command-line whenever
necessary without taking your hands off the keyboard. Just run info awk.
Your locale doesn’t use ‘.’ as decimal separator: the input/output of some core operating system tools like awk or seq depend on the system locale (https://en.wikipedia.org/wiki/Locale_(computer_software)). For example in Spanish and some other languages the decimal separator (symbol used to separate the integer and fractional part of a number), is a comma. Therefore in systems that have Spanish as their default Locale, seq will print half of unity as ‘0,5’ (instead of ‘0.5’). This can cause problems for parsing the printed numbers in other programs. You can check your current locale with the locale command. You can test your default decimal separator with this command:

```
seq 0.5 1
```

To avoid these kinds of locale-specific problems (for example another program not being able to read ‘0,5’ as half of unity), you can change the locale by setting the LANG environment variable (or the lower-level/generic LC_ALL). You can do it only for a single command (the first one below), or all commands within the running session (the second command below):

```
## Change the locale to the standard, only for this 'seq' command.
$ LANG=C seq 0.5 1

## Change the locale to the standard, for all commands after it.
$ export LANG=C
```

If you want to change it generally for all future sessions, you can put the second command in your shell’s startup file. For more on startup files, please see Section 3.3.1.2 [Installation directory], page 139.

### 2.2.6 Cosmological coverage

Having found the angular coverage of the dataset in Section 2.2.5 [Angular coverage on the sky], page 34, we can now use Gnuastro to answer a more physically motivated question: “How large is this area at different redshifts?” To get a feeling of the tangential area that this field covers at redshift 2, you can use Gnuastro’s CosmicCalculator program (Section 9.1 [CosmicCalculator], page 469). In particular, you need the tangential distance covered by 1 arc-second as raw output. Combined with the field’s area that was measured before, we can calculate the tangential distance in Mega Parsecs squared ($Mpc^2$).

```
## If your system language uses ',' (not '.') as decimal separator.
$ export LANG=C

## Print general cosmological properties at redshift 2 (for example).
$ astcosmiccal -z2

## When given a "Specific calculation" option, CosmicCalculator
## will just print that particular calculation. To see all such
## calculations, add a '--help' token to the previous command
## (under the same title). Note that with '--help', no processing
## is done, so you can always simply append it to remember
## something without modifying the command you want to run.
$ astcosmiccal -z2 --help

## Only print the "Tangential dist. covered by 1arcsec at z (kpc)".
```
## in units of kpc/arc-seconds.

$ astcosmiccal -z2 --arcsectandist

## But it's easier to use the short version of this option (which can be appended to other short options).

$ astcosmiccal -sz2

## Convert this distance to kpc^2/arcmin^2 and save in 'k'.

$ k=$(astcosmiccal -sz2 | awk '{print ($1*60)^2}')

## Re-calculate the area of the dataset in arcmin^2.

$ n=$(aststatistics flat-ir/xdf-f160w.fits --number)

$ r=$(astfits flat-ir/xdf-f160w.fits -h1 | grep CDELT1 | awk '{print $3}')

$ a=$(echo $n $r | awk '{print $1 * ($2*60)^2 }')

## Multiply 'k' and 'a' and divide by 10^6 for value in Mpc^2.

$ echo $k $a | awk '{print $1 * $2 / 1e6}'

At redshift 2, this field therefore covers approximately 1.07 Mpc^2. If you would like to see how this tangential area changes with redshift, you can use a shell loop like below.

$ for z in 0.5 1.0 1.5 2.0 2.5 3.0 3.5 4.0 4.5 5.0; do
  k=$(astcosmiccal -sz$z); 
  echo $z $k $a | awk '{print $1, ($2*60)^2 * $3 / 1e6}'; 
done

Fortunately, the shell has a useful tool/program to print a sequence of numbers that is nicely called seq. You can use it instead of typing all the different redshifts in this example. For example the loop below will calculate and print the tangential coverage of this field across a larger range of redshifts (0.1 to 5) and with finer increments of 0.1.

## If your system language uses ',' (not '.') as decimal separator.

$ export LANG=C

## The loop over the redshifts

$ for z in $(seq 0.1 0.1 5); do
  k=$(astcosmiccal -z$z --arcsectandist); 
  echo $z $k $a | awk '{print $1, ($2*60)^2 * $3 / 1e6}'; 
done

### 2.2.7 Building custom programs with the library

In Section 2.2.6 [Cosmological coverage], page 37, we repeated a certain calculation/output of a program multiple times using the shell’s for loop. This simple way repeating a calculation is great when it is only necessary once. However, if you commonly need this calculation and possibly for a larger number of redshifts at higher precision, the command above can be slow (try it out to see).

This slowness of the repeated calls to a generic program (like CosmicCalculator), is because it can have a lot of overhead on each call. To be generic and easy to operate, it has to parse the command-line and all configuration files (see Section 2.2.8 [Option management...
and configuration files], page 40) which contain human-readable characters and need a lot of pre-processing to be ready for processing by the computer. Afterwards, CosmicCalculator has to check the sanity of its inputs and check which of its many options you have asked for. All the this pre-processing takes as much time as the high-level calculation you are requesting, and it has to re-do all of these for every redshift in your loop.

To greatly speed up the processing, you can directly access the core work-horse of CosmicCalculator without all that overhead by designing your custom program for this job. Using Gnuastro’s library, you can write your own tiny program particularly designed for this exact calculation (and nothing else!). To do that, copy and paste the following C program in a file called myprogram.c.

```c
#include <math.h>
#include <stdio.h>
#include <stdlib.h>
#include <gnuastro/cosmology.h>

int main(void)
{
    double area=4.03817; /* Area of field (arcmin^2). */
    double z, adist, tandist; /* Temporary variables. */

    /* Constants from Plank 2018 (arXiv:1807.06209, Table 2) */
    double H0=67.66, olambda=0.6889, omatter=0.3111, oradiation=0;

    /* Do the same thing for all redshifts (z) between 0.1 and 5. */
    for(z=0.1; z<5; z+=0.1)
    {
        /* Calculate the angular diameter distance. */
        adist=gal_cosmology_angular_distance(z, H0, olambda,
                                              omatter, oradiation);

        /* Calculate the tangential distance of one arcsecond. */
        tandist = adist * 1000 * M_PI / 3600 / 180;

        /* Print the redshift and area. */
        printf("%-5.2f %g\n", z, pow(tandist * 60,2) * area / 1e6);
    }

    /* Tell the system that everything finished successfully. */
    return EXIT_SUCCESS;
}
```

Then run the following command to compile your program and run it.

```
$ astbuildprog myprogram.c
```

In the command above, you used Gnuastro’s BuildProgram program. Its job is to greatly simplify the compilation, linking and running of simple C programs that use Gnuastro’s
BuildProgram is designed to manage Gnuastro’s dependencies, compile and link your custom program and then run it.

Did you notice how your custom program was much faster than the repeated calls to CosmicCalculator in the previous section? You might have noticed that a new file called `myprogram` is also created in the directory. This is the compiled program that was created and run by the command above (its in binary machine code format, not human-readable any more). You can run it again to get the same results with a command like this:

```
$ ./myprogram
```

The efficiency of your custom `myprogram` compared to repeated calls to CosmicCalculator is because in the latter, the requested processing is comparable to the necessary overheads. For other programs that take large input datasets and do complicated processing on them, the overhead is usually negligible compared to the processing. In such cases, the libraries are only useful if you want a different/new processing compared to the functionalities in Gnuastro’s existing programs.

Gnuastro has a large library which is used extensively by all the programs. In other words, the library is like the skeleton of Gnuastro. For the full list of available functions classified by context, please see Section 11.3 [Gnuastro library], page 525. Gnuastro’s library and BuildProgram are created to make it easy for you to use these powerful features as you like. This gives you a high level of creativity, while also providing efficiency and robustness. Several other complete working examples (involving images and tables) of Gnuastro’s libraries can be see in Section 11.4 [Library demo programs], page 662.

But for this tutorial, let’s stop discussing the libraries at this point in and get back to Gnuastro’s already built programs which don’t need any programming. But before continuing, let’s clean up the files we don’t need any more:

```
$ rm myprogram*
```

### 2.2.8 Option management and configuration files

None of Gnuastro’s programs keep a default value internally within their code. However, when you ran CosmicCalculator only with the `-z2` option (not specifying the cosmological parameters) in Section 2.2.6 [Cosmological coverage], page 37, it completed its processing and printed results. Where did the necessary cosmological parameters (like the matter density, etc) that are necessary for its calculations come from? Fast reply: the values come from a configuration file (see Section 4.2.2 [Configuration file precedence], page 170).

CosmicCalculator is a small program with a limited set of parameters/options. Therefore, let’s use it to discuss configuration files in Gnuastro (for more, you can always see Section 4.2 [Configuration files], page 169). Configuration files are an important part of all Gnuastro’s programs, especially the ones with a large number of options, so its important to understand this part well.

Once you get comfortable with configuration files here, you can make good use of them in all Gnuastro programs (for example, NoiseChisel). For example, to do optimal detection on various datasets, you can have configuration files for different noise properties. The configuration of each program (besides its version) is vital for the reproducibility of your results, so it is important to manage them properly.

As we saw above, the full list of the options in all Gnuastro programs can be seen with the `--help` option. Try calling it with CosmicCalculator as shown below. Note how options are
grouped by context to make it easier to find your desired option. However, in each group, options are ordered alphabetically.

$ astcosmiccal --help

The options that need a value have an = sign after their long version and FLT, INT or STR for floating point numbers, integer numbers, and strings (filenames for example) respectively. All options have a long format and some have a short format (a single character), for more see Section 4.1.1.2 [Options], page 154.

When you are using a program, it is often necessary to check the value the option has just before the program starts its processing. In other words, after it has parsed the command-line options and all configuration files. You can see the values of all options that need one with the --printparams or -P option. --printparams is common to all programs (see Section 4.1.2 [Common options], page 156). In the command below, try replacing -P with --printparams to see how both do the same operation.

$ astcosmiccal -P

Let’s say you want a different Hubble constant. Try running the following command (just adding --H0=70 after the command above) to see how the Hubble constant in the output of the command above has changed.

$ astcosmiccal --H0=70

Afterwards, delete the -P and add a -z2 to see the calculations with the new cosmology (or configuration).

$ astcosmiccal --H0=70 -z2

From the output of the --help option, note how the option for Hubble constant has both short (-H) and long (--H0) formats. One final note is that the equal (=) sign is not mandatory. In the short format, the value can stick to the actual option (the short option name is just one character after-all, thus easily identifiable) and in the long format, a white-space character is also enough.

$ astcosmiccal --H0=70 -z2

$ astcosmiccal --H0 70 -z2 --arcsectandist

When an option doesn’t need a value, and has a short format (like --arcsectandist), you can easily append it before other short options. So the last command above can also be written as:

$ astcosmiccal --H0 70 -sz2

Let’s assume that in one project, you want to only use rounded cosmological parameters (H0 of 70km/s/Mpc and matter density of 0.3). You should therefore run CosmicCalculator like this:

$ astcosmiccal --H0=70 --olambda=0.7 --omatter=0.3 -z2

But having to type these extra options every time you run CosmicCalculator will be prone to errors (typos in particular), frustrating and slow. Therefore in Gnuastro, you can put all the options and their values in a “Configuration file” and tell the programs to read the option values from there.

Let’s create a configuration file... With your favorite text editor, make a file named my-cosmology.conf (or my-cosmology.txt, the suffix doesn’t matter, but a more descriptive suffix like .conf is recommended). Then put the following lines inside of it. One space
between the option value and name is enough, the values are just under each other to help in readability. Also note that you can only use long option names in configuration files.

\[
\begin{align*}
\text{H0} & \quad 70 \\
\text{olambda} & \quad 0.7 \\
\text{omatter} & \quad 0.3
\end{align*}
\]

You can now tell CosmicCalculator to read this file for option values immediately using the \texttt{--config} option as shown below. Do you see how the output of the following command corresponds to the option values in \texttt{my-cosmology.conf}, and is therefore identical to the previous command?

\[
\$ \text{astcosmiccal --config=my-cosmology.conf -z2}
\]

But still, having to type \texttt{--config=my-cosmology.conf} every time is annoying, isn't it? If you need this cosmology every time you are working in a specific directory, you can use Gnuastro's default configuration file names and avoid having to type it manually.

The default configuration files (that are checked if they exist) must be placed in the hidden \texttt{.gnuastro} sub-directory (in the same directory you are running the program). Their file name (within \texttt{.gnuastro}) must also be the same as the program's executable name. So in the case of CosmicCalculator, the default configuration file in a given directory is \texttt{.gnuastro/astcosmiccal.conf}.

Let's do this. We'll first make a directory for our custom cosmology, then build a \texttt{.gnuastro} within it. Finally, we'll copy the custom configuration file there:

\[
\begin{align*}
\$ \text{mkdir my-cosmology} \\
\$ \text{mkdir my-cosmology/.gnuastro} \\
\$ \text{mv my-cosmology.conf my-cosmology/.gnuastro/astcosmiccal.conf}
\end{align*}
\]

Once you run CosmicCalculator within \texttt{my-cosmology} (as shown below), you will see how your custom cosmology has been implemented without having to type anything extra on the command-line.

\[
\begin{align*}
\$ \text{cd my-cosmology} \\
\$ \text{astcosmiccal -P} \\
\$ \text{cd ..}
\end{align*}
\]

To further simplify the process, you can use the \texttt{--setdirconf} option. If you are already in your desired working directory, calling this option with the others will automatically write the final values (along with descriptions) in \texttt{.gnuastro/astcosmiccal.conf}. For example try the commands below:

\[
\begin{align*}
\$ \text{mkdir my-cosmology2} \\
\$ \text{cd my-cosmology2} \\
\$ \text{astcosmiccal -P} \\
\$ \text{astcosmiccal --H0 70 --olambda=0.7 --omatter=0.3 --setdirconf} \\
\$ \text{astcosmiccal -P} \\
\$ \text{cd ..}
\end{align*}
\]

Gnuastro's programs also have default configuration files for a specific user (when run in any directory). This allows you to set a special behavior every time a program is run by a specific user. Only the directory and filename differ from the above, the rest of the process is similar to before. Finally, there are also system-wide configuration files that can be used
to define the option values for all users on a system. See Section 4.2.2 [Configuration file precedence], page 170, for a more detailed discussion.

We’ll stop the discussion on configuration files here, but you can always read about them in Section 4.2 [Configuration files], page 169. Before continuing the tutorial, let’s delete the two extra directories that we don’t need any more:

```
$ rm -rf my-cosmology*
```

### 2.2.9 Warping to a new pixel grid

We are now ready to start processing the downloaded images. The XDF datasets we are using here are already aligned to the same pixel grid. However, warping to a different/matched pixel grid is commonly needed before higher-level analysis when you are using datasets from different instruments. So let’s have a look at Gnuastro’s features warping features here.

Gnuastro’s Warp program should be used for warping the pixel-grid (see Section 6.4 [Warp], page 328). For example, try rotating one of the images by 20 degrees:

```
$ astwarp flat-ir/xdf-f160w.fits --rotate=20
```

Open the output (xdx-f160w_rotated.fits) and see how it is rotated. If your final image is already aligned with RA and Dec, you can simply use the --align option and let Warp calculate the necessary rotation and apply it. For example, try aligning the rotated image back to the standard orientation (just note that because of the two rotations, the NaN parts of the image are larger now):

```
$ astwarp xdf-f160w_rotated.fits --align
```

Warp can generally be used for many kinds of pixel grid manipulation (warping), not just rotations. For example the outputs of the commands below will respectively have larger pixels (new resolution being one quarter the original resolution), get shifted by 2.8 (by sub-pixel), get a shear of 2, and be tilted (projected). Run each of them and open the output file to see the effect, they will become handy for you in the future.

```
$ astwarp flat-ir/xdf-f160w.fits --scale=0.25
$ astwarp flat-ir/xdf-f160w.fits --translate=2.8
$ astwarp flat-ir/xdf-f160w.fits --shear=0.2
$ astwarp flat-ir/xdf-f160w.fits --project=0.001,0.0005
```

If you need to do multiple warps, you can combine them in one call to Warp. For example to first rotate the image, then scale it, run this command:

```
$ astwarp flat-ir/xdf-f160w.fits --rotate=20 --scale=0.25
```

If you have multiple warps, do them all in one command. Don’t warp them in separate commands because the correlated noise will become too strong. As you see in the matrix that is printed when you run Warp, it merges all the warps into a single warping matrix (see Section 6.4.2 [Merging multiple warpings], page 331) and simply applies that (mixes the pixel values) just once. However, if you run Warp multiple times, the pixels will be mixed multiple times, creating a strong artificial blur/smoothing, or stronger correlated noise.

Recall that the merging of multiple warps is done through matrix multiplication, therefore order matters in the separate operations. At a lower level, through Warp’s --matrix option, you can directly request your desired final warp and don’t have to break it up into different warps like above (see Section 6.4.4 [Invoking Warp], page 333).
Fortunately these datasets are already aligned to the same pixel grid, so you don’t actually need the files that were just generated. You can safely delete them all with the following command. Here, you see why we put the processed outputs that we need later into a separate directory. In this way, the top directory can be used for temporary files for testing that you can simply delete with a generic command like below.

```bash
$ rm *.fits
```

### 2.2.10 NoiseChisel and Multiextension FITS files

Having completed a review of the basics in the previous sections, we are now ready to separate the signal (galaxies or stars) from the background noise in the image. We will be using the results of Section 2.2.4 [Dataset inspection and cropping], page 32, so be sure you already have them. Gnuastro has NoiseChisel for this job. But NoiseChisel’s output is a multi-extension FITS file, therefore to better understand how to use NoiseChisel, let’s take a look at multi-extension FITS files and how you can interact with them.

In the FITS format, each extension contains a separate dataset (image in this case). You can get basic information about the extensions in a FITS file with Gnuastro’s Fits program (see Section 5.1 [Fits], page 194). To start with, let’s run NoiseChisel without any options, then use Gnuastro’s FITS program to inspect the number of extensions in this file.

```bash
$ astnoisechisel flat-ir/xdf-f160w.fits
$ astfits xdf-f160w_detected.fits
```

From the output list, we see that NoiseChisel’s output contains 5 extensions and the first (counting from zero, with name NOISECHISEL-CONFIG) is empty: it has value of 0 in the last column (which shows its size). The first extension in all the outputs of Gnuastro’s programs only contains meta-data: data about/describing the datasets within (all) the output’s extensions. This is recommended by the FITS standard, see Section 5.1 [Fits], page 194, for more. In the case of Gnuastro’s programs, this generic zero-th/meta-data extension (for the whole file) contains all the configuration options of the program that created the file.

The second extension of NoiseChisel’s output (numbered 1, named INPUT-NO-SKY) is the Sky-subtracted input that you provided. The third (DETECTIONS) is NoiseChisel’s main output which is a binary image with only two possible values for all pixels: 0 for noise and 1 for signal. Since it only has two values, to avoid taking too much space on your computer, its numeric datatype an unsigned 8-bit integer (or uint8)\(^{14}\). The fourth and fifth (SKY and SKY_STD) extensions, have the Sky and its standard deviation values for the input on a tile grid and were calculated over the undetected regions (for more on the importance of the Sky value, see Section 7.1.4 [Sky value], page 344).

Metadata regarding how the analysis was done (or a dataset was created) is very important for higher-level analysis and reproducibility. Therefore, let’s first take a closer look at the NOISECHISEL-CONFIG extension. If you specify a special header in the FITS file, Gnuastro’s Fits program will print the header keywords (metadata) of that extension. You can either specify the HDU/extension counter (starting from 0), or name. Therefore, the two commands below are identical for this file:

```bash
$ astfits xdf-f160w_detected.fits -h0
```

---

\(^{14}\) To learn more about numeric data types see Section 4.5 [Numeric data types], page 178.
Chapter 2: Tutorials

$ astfits xdf-f160w_detected.fits -hNOISECHISEL-CONFIG

The first group of FITS header keywords are standard keywords (containing the SIMPLE and BITPIX keywords the first empty line). They are required by the FITS standard and must be present in any FITS extension. The second group contains the input file and all the options with their values in that run of NoiseChisel. Finally, the last group contains the date and version information of Gnuastro and its dependencies. The “versions and date” group of keywords are present in all Gnuastro’s FITS extension outputs, for more see Section 4.10 [Output FITS files], page 191.

Note that if a keyword name is larger than 8 characters, it is preceded by a HIERARCH keyword and that all keyword names are in capital letters. Therefore, if you want to see only one keyword’s value by feeding the output to Grep, you should ask Grep to ignore case with its -i option (short name for --ignore-case). For example, below we’ll check the value to the --snminarea option, note how we don’t need Grep’s -i option when it is fed with astnoisechisel -P since it is already in small-caps there. The extra white spaces in the first command are only to help in readability, you can ignore them when typing.

$ astnoisechisel -P | grep snminarea
$ astfits xdf-f160w_detected.fits -h0 | grep -i snminarea

The metadata (that is stored in the output) can later be used to exactly reproduce/understand your result, even if you have lost/forgot the command you used to create the file. This feature is present in all of Gnuastro’s programs, not just NoiseChisel.

Let’s continue with the extensions in NoiseChisel’s output that contain a dataset by visually inspecting them (here, we’ll use SAO DS9). Since the file contains multiple related extensions, the easiest way to view all of them in DS9 is to open the file as a “Multi-extension data cube” with the -mecube option as shown below.

$ ds9 -mecube xdf-f160w_detected.fits -zscale -zoom to fit

To simplify viewing multi-extension FITS images in SAO DS9 (or tables in TOPCAT) and avoid such long commands, Gnuastro has the astscript-fits-view. It is just a simple wrapper to call them with some useful customizations depending on your input (see Section 10.4 [Viewing FITS file contents with DS9 or TOPCAT], page 494). It also comes with a .desktop file (usable in GNOME or KDE graphic user interfaces or GUIs) to enable easy double-clicking on any number of FITS files to open the respective program (very useful when navigating in your graphic environment):

$ astscript-fits-view xdf-f160w_detected.fits

A “cube” window opens along with DS9’s main window. The buttons and horizontal scroll bar in this small new window can be used to navigate between the extensions. In this mode, all DS9’s settings (for example zoom or color-bar) will be identical between the extensions. Try zooming into one part and flipping through the extensions to see how the galaxies were detected along with the Sky and Sky standard deviation values for that region. Just have in mind that NoiseChisel’s job is only detection (separating signal from noise), We’ll do segmentation on this result later to find the individual galaxies/peaks over the detected pixels.

---

15 You can configure your graphic user interface to open DS9 in multi-extension cube mode by default when using the GUI (double clicking on the file)
Each HDU/extension in a FITS file is an independent dataset (image or table) which you can delete from the FITS file, or copy/cut to another file. For example, with the command below, you can copy NoiseChisel’s DETECTIONS HDU/extension to another file:

```
$ astfits xdf-f160w_detected.fits --copy=DETECTIONS -odetections.fits
```

There are similar options to conveniently cut (--cut, copy, then remove from the input) or delete (--remove) HDUs from a FITS file also. See Section 5.1.1.1 [HDU information and manipulation], page 197, for more.

### 2.2.11 NoiseChisel optimization for detection

In Section 2.2.10 [NoiseChisel and Multiextension FITS files], page 44, we ran NoiseChisel and reviewed NoiseChisel’s output format. Now that you have a better feeling for multi-extension FITS files, let’s optimize NoiseChisel for this particular dataset.

One good way to see if you have missed any signal (small galaxies, or the wings of brighter galaxies) is to mask all the detected pixels and inspect the noise pixels. For this, you can use Gnuastro’s Arithmetic program (in particular its `where` operator, see Section 6.2.3 [Arithmetic operators], page 276). The command below will produce `mask-det.fits`. In it, all the pixels in the **INPUT-NO-SKY** extension that are flagged 1 in the **DETECTIONS** extension (dominated by signal, not noise) will be set to NaN.

Since the various extensions are in the same file, for each dataset we need the file and extension name. To make the command easier to read/write/understand, let’s use shell variables: ‘in’ will be used for the Sky-subtracted input image and ‘det’ will be used for the detection map. Recall that a shell variable’s value can be retrieved by adding a `$` before its name, also note that the double quotations are necessary when we have white-space characters in a variable name (like this case).

```
$ in="xdf-f160w_detected.fits -hINPUT-NO-SKY"
$ det="xdf-f160w_detected.fits -hDETECTIONS"
$ astarithmetic $in $det nan where --output=mask-det.fits
```

To invert the result (only keep the detected pixels), you can flip the detection map (from 0 to 1 and vice-versa) by adding a ‘not’ after the second `$det`:

```
$ astarithmetic $in $det not nan where --output=mask-sky.fits
```

Look again at the DETECTIONS extension, in particular the long worm-like structure around\(^\text{16}\) pixel 1650 (X) and 1470 (Y). These types of long wiggly structures show that we have dug too deep into the noise, and are a signature of correlated noise. Correlated noise is created when we warp (for example rotate) individual exposures (that are each slightly offset compared to each other) into the same pixel grid before adding them into one deeper image. During the warping, nearby pixels are mixed and the effect of this mixing on the noise (which is in every pixel) is called “correlated noise”. Correlated noise is a form of convolution and it slightly smooths the image.

---
16 To find a particular coordinate easily in DS9, you can do this: Click on the “Edit” menu, and select “Region”. Then click on any random part of the image to see a circle show up in that location (this is the “region”). Double-click on the region and a “Circle” window will open. If you have celestial coordinates, keep the default “fk5” in the scroll-down menu after the “Center”. But if you have pixel/image coordinates, click on the “fk5” and select “Image”. Now you can set the “Center” coordinates of the region (1650 and 1470 in this case) by manually typing them in the two boxes in front of “Center”. Finally, when everything is ready, click on the “Apply” button and your region will go over your requested coordinates. You can zoom out (to see the whole image) and visually find it.
In terms of the number of exposures (and thus correlated noise), the XDF dataset is by no means an ordinary dataset. Therefore the default parameters need to be slightly customized. It is the result of warping and adding roughly 80 separate exposures which can create strong correlated noise/smoothing. In common surveys the number of exposures is usually 10 or less. See Figure 2 of Akhlaghi [2019] (https://arxiv.org/abs/1909.11230) and the discussion on `--detgrowquant` there for more on how NoiseChisel “grow”s the detected objects and the patterns caused by correlated noise.

Let’s tweak NoiseChisel’s configuration a little to get a better result on this dataset. Don’t forget that “Good statistical analysis is not a purely routine matter, and generally calls for more than one pass through the computer” (Anscombe 1973, see Section 1.2 [Gnuastro manifesto: Science and its tools], page 2). A good scientist must have a good understanding of her tools to make a meaningful analysis. So don’t hesitate in playing with the default configuration and reviewing the manual when you have a new dataset (from a new instrument) in front of you. Robust data analysis is an art, therefore a good scientist must first be a good artist. Once you have found the good configuration for that particular noise pattern (instrument) you can safely use it for all new data that have a similar noise pattern.

NoiseChisel can produce “Check images” to help you visualize and inspect how each step is done. You can see all the check images it can produce with this command.

```
$ astnoisechisel --help | grep check
```

Let’s check the overall detection process to get a better feeling of what NoiseChisel is doing with the following command. To learn the details of NoiseChisel in more detail, please see Section 7.2 [NoiseChisel], page 362, Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664) and Akhlaghi [2019] (https://arxiv.org/abs/1909.11230).

```
$ astnoisechisel flat-ir/xdf-f160w.fits --checkdetection
```

The check images/tables are also multi-extension FITS files. As you saw from the command above, when check datasets are requested, NoiseChisel won’t go to the end. It will abort as soon as all the extensions of the check image are ready. Please list the extensions of the output with `astfits` and then opening it with `ds9` as we done above. If you have read the paper, you will see why there are so many extensions in the check image.

```
$ astfits xdf-f160w_detcheck.fits
$ ds9 -mecube xdf-f160w_detcheck.fits -zscale -zoom to fit
```

In order to understand the parameters and their biases (especially as you are starting to use Gnuastro, or running it a new dataset), it is strongly encouraged to play with the different parameters and use the respective check images to see which step is affected by your changes and how, for example see Section 2.3 [Detecting large extended targets], page 76.

Let’s focus on one step: the `OPENED_AND_LABELED` extension shows the initial detection step of NoiseChisel. We see the seeds of that correlated noise structure with many small detections (a relatively early stage in the processing). Such connections at the lowest surface brightness limits usually occur when the dataset is too smoothed, the threshold is too low, or the final “growth” is too much.

As you see from the 2nd (`CONVOLVED`) extension, the first operation that NoiseChisel does on the data is to slightly smooth it. However, the natural correlated noise of this dataset is already one level of artificial smoothing, so further smoothing it with the default kernel may be the culprit. To see the effect, let’s use a sharper kernel as a first step to convolve/smooth the input.
By default NoiseChisel uses a Gaussian with full-width-half-maximum (FWHM) of 2 pixels. We can use Gnuastro’s MakeProfiles to build a kernel with FWHM of 1.5 pixel (truncated at 5 times the FWHM, like the default) using the following command. MakeProfiles is a powerful tool to build any number of mock profiles on one image or independently, to learn more of its features and capabilities, see Section 8.1 [MakeProfiles], page 440.

```
$ astmkprof --kernel=gaussian,1.5,5 --oversample=1
```

Please open the output `kernel.fits` and have a look (it is very small and sharp). We can now tell NoiseChisel to use this instead of the default kernel with the following command (we’ll keep the `--checkdetection` to continue checking the detection steps):

```
$ astnoisechisel flat-ir/xdf-f160w.fits --kernel=kernel.fits --checkdetection
```

Open the output `xdf-f160w_detcheck.fits` as a multi-extension FITS file and go to the last extension (DETECTIONS-FINAL, it is the same pixels as the final NoiseChisel output without `--checkdetections`). Look again at that position mentioned above (1650,1470), you see that the long wiggly structure is gone. This shows we are making progress :-).

Looking at the new OPENED_AND_LABELED extension, we see that the thin connections between smaller peaks has now significantly decreased. Going two extensions/steps ahead (in the first HOLES-FILLED), you can see that during the process of finding false pseudo-detections, too many holes have been filled: do you see how the many of the brighter galaxies are connected? At this stage all holes are filled, irrespective of their size.

Try looking two extensions ahead (in the first PSEUDOS-FOR-SN), you can see that there aren’t too many pseudo-detections because of all those extended filled holes. If you look closely, you can see the number of pseudo-detections in the printed outputs of NoiseChisel (around 6400). This is another side-effect of correlated noise. To address it, we should slightly increase the pseudo-detection threshold (before changing `--dthresh`, run with `-P` to see the default value):

```
$ astnoisechisel flat-ir/xdf-f160w.fits --kernel=kernel.fits --dthresh=0.1 --checkdetection
```

Before visually inspecting the check image, you can already see the effect of this small change in NoiseChisel’s command-line output: notice how the number of pseudo-detections has increased to more than 7100! Open the check image now and have a look, you can see how the pseudo-detections are distributed much more evenly in the blank sky regions of the PSEUDOS-FOR-SN extension.

---

**Maximize the number of pseudo-detections:** When using NoiseChisel on datasets with a new noise-pattern (for example going to a Radio astronomy image, or a shallow ground-based image), play with `--dthresh` until you get a maximal number of pseudo-detections: the total number of pseudo-detections is printed on the command-line when you run NoiseChisel, you don’t even need to open a FITS viewer.

In this particular case, try `--dthresh=0.2` and you will see that the total printed number decreases to around 6700 (recall that with `--dthresh=0.1`, it was roughly 7100). So for this type of very deep HST images, we should set `--dthresh=0.1`.

---

As discussed in Section 3.1.5 of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664), the signal-to-noise ratio of pseudo-detections are critical to identify-
ing/removing false detections. For an optimal detection they are very important to get right (where you want to detect the faintest and smallest objects in the image successfully). Let’s have a look at their signal-to-noise distribution with --checksn.

$ astnoisechisel flat-ir/xdf-f160w.fits --kernel=kernel.fits \ 
   --dthresh=0.1 --checkdetection --checksn

The output (xdf-f160w_detsn.fits) contains two extensions for the pseudo-detections containing two-column tables over the undetected (SKY_PSEUDODET_SN) regions and those over detections (DET_PSEUDODET_SN). With the first command below you can see the HDUs of this file, and with the second you can see the information of the table in the first HDU (which is the default when you don’t use --hdu):

$ astfits xdf-f160w_detsn.fits
$ asttable xdf-f160w_detsn.fits -i

You can see the table columns with the first command below and get a feeling of the signal-to-noise value distribution with the second command (the two Table and Statistics programs will be discussed later in the tutorial):

$ asttable xdf-f160w_detsn.fits -hSKY_PSEUDODET_SN
$ aststatistics xdf-f160w_detsn.fits -hSKY_PSEUDODET_SN -c2
... [output truncated] ...

Histogram:

| * |
| *** |
| ***** |
| ********* |
| ********** |
| ************* |
| ***************** |
| ******************** |
| ************************** |
| ******************************************************* *
| ** |
|----------------------------------------------------------------------

The correlated noise is again visible in the signal-to-noise distribution of sky pseudo-detections! Do you see how skewed this distribution is? In an image with less correlated noise, this distribution would be much more symmetric. A small change in the quantile will translate into a big change in the S/N value. For example see the difference between the three 0.99, 0.95 and 0.90 quantiles with this command:

$ aststatistics xdf-f160w_detsn.fits -hSKY_PSEUDODET_SN -c2 \ 
   --quantile=0.99 --quantile=0.95 --quantile=0.90

We get a change of almost 2 units (which is very significant). If you run NoiseChisel with -P, you’ll see the default signal-to-noise quantile --snquant is 0.99. In effect with this option you specify the purity level you want (contamination by false detections). With the aststatistics command above, you see that a small number of extra false detections (impurity) in the final result causes a big change in completeness (you can detect more lower signal-to-noise true detections). So let’s loosen-up our desired purity level, remove
the check-image options, and then mask the detected pixels like before to see if we have missed anything.

```
$ astnoisechisel flat-ir/xdf-f160w.fits --kernel=kernel.fits \
   --dthresh=0.1 --snquant=0.95
$ in="xdf-f160w_detected.fits -hINPUT-NO-SKY"
$ det="xdf-f160w_detected.fits -hDETECTIONS"
$ astarithmetic $in $det nan where --output=mask-det.fits
```

Overall it seems good, but if you play a little with the color-bar and look closer in the noise, you’ll see a few very sharp, but faint, objects that have not been detected. For example the object around pixel (456, 1662). Despite its high valued pixels, this object was lost because erosion ignores the precise pixel values. Loosing small/sharp objects like this only happens for under-sampled datasets like HST (where the pixel size is larger than the point spread function FWHM). So this won’t happen on ground-based images.

To address this problem of sharp objects, we can use NoiseChisel’s `--noerodequant` option. All pixels above this quantile will not be eroded, thus allowing us to preserve small/sharp objects (that cover a small area, but have a lot of signal in it). Check its default value, then run NoiseChisel like below and make the mask again.

```
$ astnoisechisel flat-ir/xdf-f160w.fits --kernel=kernel.fits \
   --noerodequant=0.95 --dthresh=0.1 --snquant=0.95
```

This seems to be fine and the object above is now detected. We’ll stop the configuration here, but please feel free to keep looking into the data to see if you can improve it even more.

Once you have found the proper customization for the type of images you will be using you don’t need to change them any more. The same configuration can be used for any dataset that has been similarly produced (and has a similar noise pattern). But entering all these options on every call to NoiseChisel is annoying and prone to bugs (mistakenly typing the wrong value for example). To simply things, we’ll make a configuration file in a visible `config` directory. Then we’ll define the hidden `.gnuastro` directory (that all Gnuastro’s programs will look into for configuration files) as a symbolic link to the `config` directory. Finally, we’ll write the finalized values of the options into NoiseChisel’s standard configuration file within that directory. We’ll also put the kernel in a separate directory to keep the top directory clean of any files we later need.

```
$ mkdir kernel config
$ ln -s config/ .gnuastro
$ mv kernel.fits kernel/noisechisel.fits
$ echo "kernel kernel/noisechisel.fits" > config/astnoisechisel.conf
$ echo "noerodequant 0.95" >> config/astnoisechisel.conf
$ echo "dthresh 0.1" >> config/astnoisechisel.conf
$ echo "snquant 0.95" >> config/astnoisechisel.conf
```

We are now ready to finally run NoiseChisel on the three filters and keep the output in a dedicated directory (which we’ll call `nc` for simplicity).

```
$ rm *.fits
$ mkdir nc
$ for f in f105w f125w f160w; do \
   astnoisechisel flat-ir/xdf-$f.fits --output=nc/xdf-$f.fits; \
```
done

2.2.12 NoiseChisel optimization for storage

As we showed before (in Section 2.2.10 [NoiseChisel and Multiextension FITS files], page 44), NoiseChisel’s output is a multi-extension FITS file with several images the same size as the input. As the input datasets get larger this output can become hard to manage and waste a lot of storage space. Fortunately there is a solution to this problem (which is also useful for Segment’s outputs).

In this small section we’ll take a short detour to show this feature. Please note that the outputs generated here are not needed for the rest of the tutorial. But first, let’s have a look at the contents/HDUs and volume of NoiseChisel’s output from Section 2.2.11 [NoiseChisel optimization for detection], page 46, (fast answer, its larger than 100 mega-bytes):

```bash
$ astfits nc/xdf-f160w.fits
$ ls -lh nc/xdf-f160w.fits
```

Two options can drastically decrease NoiseChisel’s output file size: 1) With the `--rawoutput` option, NoiseChisel won’t create a Sky-subtracted input. After all, it is redundant: you can always generate it by subtracting the SKY extension from the input image (which you have in your database) using the Arithmetic program. 2) With the `--oneelempertile`, you can tell NoiseChisel to store its Sky and Sky standard deviation results with one pixel per tile (instead of many pixels per tile). So let’s run NoiseChisel with these options, then have another look at the HDUs and the over-all file size:

```bash
$ astnoisechisel flat-ir/xdf-f160w.fits --oneelempertile --rawoutput \
   --output=nc-for-storage.fits
$ astfits nc-for-storage.fits
$ ls -lh nc-for-storage.fits
```

See how `nc-for-storage.fits` has four HDUs, while `nc/xdf-f160w.fits` had five HDUs? As explained above, the missing extension is INPUT-NO-SKY. Also, look at the sizes of the SKY and SKY_STD HDUs, unlike before, they aren’t the same size as DETECTIONS, they only have one pixel for each tile (group of pixels in raw input). Finally, you see that `nc-for-storage.fits` is just under 8 mega byes (while `nc/xdf-f160w.fits` was 100 mega bytes)!

But were are not finished! You can even be more efficient in storage, archival or transferring NoiseChisel’s output by compressing this file. Try the command below to see how NoiseChisel’s output has now shrunk to about 250 kilo-byes while keeping all the necessary information as the original 100 mega-byte output.

```bash
$ gzip --best nc-for-storage.fits
$ ls -lh nc-for-storage.fits.gz
```

We can get this wonderful level of compression because NoiseChisel’s output is binary with only two values: 0 and 1. Compression algorithms are highly optimized in such scenarios.

You can open `nc-for-storage.fits.gz` directly in SAO DS9 or feed it to any of Gnuastro’s programs without having to decompress it. Higher-level programs that take NoiseChisel’s output (for example Segment or MakeCatalog) can also deal with this compressed image where the Sky and its Standard deviation are one pixel-per-tile. You just have to give the “values” image as a separate option, for more, see Section 7.3 [Segment], page 381, and Section 7.4 [MakeCatalog], page 392.
Segment (the program we will introduce in the next section for identifying sub-structure), also has similar features to optimize its output for storage. Since this file was only created for a fast detour demonstration, let’s keep our top directory clean and move to the next step:

```
rm nc-for-storage.fits.gz
```

### 2.2.13 Segmentation and making a catalog

The main output of NoiseChisel is the binary detection map ([DETECTIONS extension, see Section 2.2.11 [NoiseChisel optimization for detection], page 46](#)), which only has two values of 1 or 0. This is useful when studying the noise or background properties, but hardly of any use when you actually want to study the targets/galaxies in the image, especially in such a deep field where almost everything is connected. To find the galaxies over the detections, we’ll use Gnuastro’s Section 7.3 ([Segment], page 381), [program](#):

```bash
$ mkdir seg
$ astsegment nc/xdf-f160w.fits -oseg/xdf-f160w.fits
$ astsegment nc/xdf-f125w.fits -oseg/xdf-f125w.fits
$ astsegment nc/xdf-f105w.fits -oseg/xdf-f105w.fits
```

Segment’s operation is very much like NoiseChisel (in fact, prior to version 0.6, it was part of NoiseChisel). For example the output is a multi-extension FITS file, it has check images and uses the undetected regions as a reference. Please have a look at Segment’s multi-extension output with [ds9](#) to get a good feeling of what it has done.

```bash
$ ds9 -mecube seg/xdf-f160w.fits -zscale -zoom to fit
```

Like NoiseChisel, the first extension is the input. The [CLUMPS extension shows the true “clumps” with values that are ≥ 1, and the diffuse regions labeled as −1](#). Please flip between the first extension and the clumps extension and zoom-in on some of the clumps to get a feeling of what they are. In the [OBJECTS extension, we see that the large detections of NoiseChisel (that may have contained many galaxies) are now broken up into separate labels.](#) Play with the color-bar and hover your mouse of the various detections to see their different labels.

The clumps are not affected by the hard-to-deblend and low signal-to-noise diffuse regions, they are more robust for calculating the colors (compared to objects). From this step onward, we’ll continue with clumps.

Having localized the regions of interest in the dataset, we are ready to do measurements on them with Section 7.4 ([MakeCatalog], page 392). MakeCatalog is specialized and optimized for doing measurements over labeled regions of an image. In other words, through MakeCatalog, you can “reduce” an image to a table (catalog of certain properties of objects in the image). Each requested measurement (over each label) will be given a column in the output table. To see the full set of available measurements run it with [--help](#) like below (and scroll up), note that measurements are classified by context.

```bash
$ astmkcatalog --help
```

So let’s select the properties we want to measure in this tutorial. First of all, we need to know which measurement belongs to which object or clump, so we’ll start with the **--ids** (read as: **IDs**). We also want to measure (in this order) the Right Ascension (with

---

This option is plural because we need two ID columns for identifying “clumps” in the clumps catalog/table: the first column will be the ID of the host “object”, and the second one will be the ID of the
Declination (--dec), magnitude (--magnitude), and signal-to-noise ratio (--sn) of the objects and clumps. Furthermore, as mentioned above, we also want measurements on clumps, so we also need to call --clumpscat. The following command will make these measurements on Segment’s F160W output and write them in a catalog for each object and clump in a FITS table.

```
$ mkdir cat
$ astmkcatalog seg/xdf-f160w.fits --ids --ra --dec --magnitude --sn --zeropoint=25.94 --clumpscat --output=cat/xdf-f160w.fits
```

From the printed statements on the command-line, you see that MakeCatalog read all the extensions in Segment’s output for the various measurements it needed. To calculate colors, we also need magnitude measurements on the other filters. So let’s repeat the command above on them, just changing the file names and zero point (which we got from the XDF survey web page):

```
$ astmkcatalog seg/xdf-f125w.fits --ids --ra --dec --magnitude --sn --zeropoint=26.23 --clumpscat --output=cat/xdf-f125w.fits

$ astmkcatalog seg/xdf-f105w.fits --ids --ra --dec --magnitude --sn --zeropoint=26.27 --clumpscat --output=cat/xdf-f105w.fits
```

However, the galaxy properties might differ between the filters (which is the whole purpose behind observing in different filters!). Also, the noise properties and depth of the datasets differ. You can see the effect of these factors in the resulting clump catalogs, with Gnuastro’s Table program. We’ll go deep into working with tables in the next section, but in summary: the -i option will print information about the columns and number of rows. To see the column values, just remove the -i option. In the output of each command below, look at the Number of rows:; and note that they are different.

```
$ asttable cat/xdf-f105w.fits -hCLUMPS -i
$ asttable cat/xdf-f125w.fits -hCLUMPS -i
$ asttable cat/xdf-f160w.fits -hCLUMPS -i
```

Matching the catalogs is possible (for example with Section 7.5 [Match], page 429). However, the measurements of each column are also done on different pixels: the clump labels can/will differ from one filter to another for one object. Please open them and focus on one object to see for your self. This can bias the result, if you match catalogs.

An accurate color calculation can only be done when magnitudes are measured from the same pixels on all images and this can be done easily with MakeCatalog. In fact this is one of the reasons that NoiseChisel or Segment don’t generate a catalog like most other detection/segmentation software. This gives you the freedom of selecting the pixels for measurement in any way you like (from other filters, other software, manually, and etc). Fortunately in these images, the Point spread function (PSF) is very similar, allowing us to use a single labeled image output for all filters\(^\text{18}\).

The F160W image is deeper, thus providing better detection/segmentation, and redder, thus observing smaller/older stars and representing more of the mass in the galaxies. We will

\(^{18}\) When the PSFs between two images differ largely, you would have to PSF-match the images before using the same pixels for measurements.
thus use the F160W filter as a reference and use its segment labels to identify which pixels to use for which objects/clumps. But we will do the measurements on the sky-subtracted F105W and F125W images (using MakeCatalog’s `--valuesfile` option) as shown below: Notice that the only difference between these calls and the call to generate the raw F160W catalog (excluding the zero point and the output name) is the `--valuesfile`.

```
$ astmkcatalog seg/xdf-f160w.fits --ids --ra --dec --magnitude --sn \
   --valuesfile=nc/xdf-f125w.fits --zeropoint=26.23 \
   --clumpscat --output=cat/xdf-f125w-on-f160w-lab.fits

$ astmkcatalog seg/xdf-f160w.fits --ids --ra --dec --magnitude --sn \
   --valuesfile=nc/xdf-f105w.fits --zeropoint=26.27 \
   --clumpscat --output=cat/xdf-f105w-on-f160w-lab.fits
```

After running the commands above, look into what MakeCatalog printed on the command-line. You can see that (as requested) the object and clump pixel labels in both were taken from the respective extensions in `seg/xdf-f160w.fits`. However, the pixel values and pixel Sky standard deviation were respectively taken from `nc/xdf-f105w.fits` and `nc/xdf-f125w.fits`. Since we used the same labeled image on all filters, the number of rows in both catalogs are now identical. Let’s have a look:

```
$ asttable cat/xdf-f105w-on-f160w-lab.fits -hCLUMPS -i
$ asttable cat/xdf-f125w-on-f160w-lab.fits -hCLUMPS -i
$ asttable cat/xdf-f160w.fits -hCLUMPS -i
```

Finally, MakeCatalog also does basic calculations on the full dataset (independent of each labeled region but related to whole data), for example pixel area or per-pixel surface brightness limit. They are stored as keywords in the FITS headers (or lines starting with # in plain text). You can see them with this command (for more, see Section 2.3.4 [Image surface brightness limit], page 88, in the next tutorial):

```
$ astfits cat/xdf-f160w.fits -h1
```

### 2.2.14 Measuring the dataset limits

In Section 2.2.13 [Segmentation and making a catalog], page 52, we created a catalog of the different objects with the image. Before measuring colors, or doing any other kind of analysis on the catalogs (and detected objects), it is very important to understand the limitations of the dataset. Without understanding the limitations of your dataset, you cannot make any physical interpretation of your results. The theory behind the calculations discussed here is thoroughly introduced in Section 7.4.3 [Quantifying measurement limits], page 398.

For example, with the command below, let’s sort all the detected clumps in the image by magnitude, and have a look at the RA, Declination, magnitude and S/N columns after sorting:

```
$ asttable cat/xdf-f160w.fits -hclumps -cmagnitude,sn -cra,dec \
   --sort=magnitude --noblank=magnitude
```

As you see, we have clumps with a total magnitude of almost 32! Let’s have a look at all of those with a magnitude between 31 and 32:

```
$ asttable cat/xdf-f160w.fits -hclumps -cmagnitude,sn \
   --range=magnitude,31:32 \
   | astscript-ds9-region -c1,2 --radius=0.5 \
```
There actually does seem to be a true peak under the selected regions, but as you see, they are very small, diffuse and noisy. How reliable are the measured magnitudes? Using the S/N column from the first command above, you can see that such objects only have a signal to noise of about 1.5.

This brings us to the first method of quantifying your dataset’s magnitude limit, which is also sometimes called detection limit (see Section 7.4.3.5 [Magnitude limit of image], page 402). To estimate the 5σ detection limit of your dataset, you simply report the median magnitude of the objects that have a signal to noise of (approximately) five. This is very easy to calculate with the command below:

```
$ asttable cat/xdf-f160w.fits -hclumps --range=sn,4.8:5.2 -cmagnitude \
| aststatistics --median
```

```
29.9627
```

But this is unreasonably faint! So let’s have a look at these objects, to get a feeling of what these clump looks like:

```
$ asttable cat/xdf-f160w.fits -hclumps --range=sn,4.8:5.2 \
    -cra,dec,magnitude \
| astscript-ds9-region -c1,2 --namecol=3 \
  --width=2 --radius=0.5 \
  --command="ds9 -mecube seg/xdf-f160w.fits -zsacle"
```

The number you see on top of each region is the clump’s magnitude. Please go over the objects and have a close look at them! It is very important to have a feeling of what your dataset looks like, and how to interpret the numbers to associate an image with them.

Generally, they look very small and seem to be much fainter than what you would expect to be 5σ. The main issue is that MakeCatalog uses individual pixel measurements of signal and noise to estimate this. However, during the reduction many exposures are co-added and stacked, mixing the pixels like a small convolution.

A more realistic way to estimate the significance of the detection is to take its footprint, randomly place it in thousands of undetected regions of the image and use that distribution as a reference. This is technically known as upper-limit measurements. For a full discussion, see Section 7.4.3.4 [Upper limit magnitude of each detection], page 400.

Since its for each separate object, the upper-limit measurements should be requested as extra columns in MakeCatalog’s output. For example with the command below, let’s generate a new catalog of the F160W filter, but with two extra columns compared to the one in cat/: the upper-limit magnitude and the upper-limit multiple of sigma.

```
$ astmkcatalog seg/xdf-f160w.fits --ids --ra --dec --magnitude --sn \
    --zeropoint=25.94 --clumpscat --upnsigma=3 \ 
    --upperlimitmag --upperlimitsigma \ 
    --output=xdf-f160w.fits
```

Let’s compare the upper-limit magnitude with the measured magnitude of each clump:

```
$ asttable xdf-f160w.fits -hclumps -cmagnitude,upperlimit_mag
```

As you see, in almost all of the cases, the measured magnitude is sufficiently higher than the upper-limit magnitude. For these objects, the clump seems to have sufficiently higher
brightness than the noisy background. Let’s use Table’s Section 5.3.1 [Column arithmetic], page 230, to find only those that have a positive difference:

```bash
$ asttable xdf-f160w.fits -hclumps -cra,dec --noblankend=3 \
-c 'arith magnitude upperlimit_mag - set-d d d 0 lt nan where'
```

From more than 3500 clumps, this command only gave 177 rows! Let’s have a look at them:

```bash
$ asttable xdf-f160w.fits -hclumps -cra,dec --noblankend=3 \
-c 'arith magnitude upperlimit_mag - set-d d d 0 lt nan where' \
| astscript-ds9-region -c1,2 --namecol=3 --width=2 \
--radius=0.5 \
--command="ds9 -mecube seg/xdf-f160w.fits -zscale"
```

You see that they are all extremely faint and diffuse/small peaks. Therefore, if an object’s magnitude is fainter than its upper-limit magnitude, you shouldn’t use the magnitude: it is not accurate! You should use the upper-limit magnitude instead (with an arrow in your plots to mark which ones are upper-limits).

But the main point (in relation to the magnitude limit) with the upper-limit, is the UPPERLIMIT_SIGMA column. you can think of this as a realistic S/N for extremely faint/diffuse/small objects. The raw S/N column is simply calculated on a pixel-by-pixel basis, however, the upper-limit sigma is produced by actually taking the label’s footprint, and randomly placing it thousands of time over un-detected parts of the image and measuring the brightness of the sky. The clump’s brightness is then divided by the standard deviation of the resulting distribution to give you exactly how significant it is (accounting for inter-pixel issues like correlated noise, which are strong in this dataset). You can actually compare the two values with the command below:

```bash
$ asttable xdf-f160w.fits -hclumps -csn,upperlimit_sigma
```

As you see, the second column (upper-limit sigma) is almost always less than the S/N. This clearly shows the effect of correlated noise! If you now use this column as the reference for deriving the magnitude limit, you will see that it will shift by almost 0.5 magnitudes brighter and is now more reasonable:

```bash
$ asttable xdf-f160w.fits -hclumps --range=upperlimit_sigma,4.8:5.2 \
-cmagnitude | aststatistics --median
```

29.6257

We see that the $5\sigma$ detection limit is $\sim 29.6$! This is extremely deep! For example in the Legacy Survey\(^{19}\), the $5\sigma$ detection limit for point sources is approximately 24.5 (5 magnitudes shallower than this image).

An important caveat in this simple calculation is that we should only be looking at point-like objects, not simply everything. This is because the shape or radial slope of the profile has an important effect on this measurement: at the same total magnitude, a sharper object will have a higher S/N. To be more precise, we should first perform star-galaxy separation, then do this only the objects classified as stars. A crude, first-order, method is to use the --axisratio option so MakeCatalog also measures the axis ratio, then call Table with --range=sn,4.8:5.2 and --range=axis_ratio,0.95:1 (in one command). Before continuing, let’s remove this temporarily produced catalog:

```bash
$ rm xdf-f160w.fits
```

\(^{19}\) https://www.legacysurvey.org/dr9/description
Another measure of the dataset’s limit is the completeness limit (Section 7.4.3.3 [Completeness limit of each detection], page 400). This is necessary when you are looking at populations of objects over the image. You want to know until what magnitude you can be sure that you have detected an object (if it was present). As described in Section 7.4.3.3 [Completeness limit of each detection], page 400, the best way to do this is with mock images. But a crude, first order result can be obtained from the actual image: by simply plotting the histogram of the magnitudes:

$ aststatistics cat/xdf-f160w.fits -hclumps -cmagnitude

Histogram:

This plot (the histogram of magnitudes; where fainter magnitudes are towards the right) is technically called the dataset’s number count plot. You see that the number of objects increases with magnitude as the magnitudes get fainter (to the right). However, beyond a certain magnitude, you see it becomes flat, and soon afterwards, the numbers suddenly drop.

Once you have your catalog, you can easily find this point with the two commands below. First we generate a histogram with fewer bins (to have more numbers in each bin). We then use AWK to find the magnitude bin where the number of points decrease compared to the previous bin. But we only do this for bins that have more than 50 items (to avoid scatter in the bright end). Finally, in Statistics, we have manually set the magnitude range and number of bins so each bin is roughly 0.5 magnitudes thick (with --greaterequal=20, --lessthan=32 and --numbins=24)

$ aststatistics cat/xdf-f160w.fits -hclumps -cmagnitude --histogram --greaterequal=20 --lessthan=32 --numbins=24 \
   --output=f160w-hist.txt

$ asttable f160w-hist.txt \
   | awk '{if ($2>50 && $2<prev{print $1; exit} {prev=$2}}'

29.10874136289

Therefore, to first order (and very crudely!) we can say that if an object is in our field of view and has a magnitude of 29.1 or brighter, we can be highly confident that we have detected it. But before continuing, let’s clean up behind ourselves:

$ rm f160w-hist.txt
Another important limiting parameter in a processed dataset is the surface brightness limit (Section 7.4.3.6 [Surface brightness limit of image], page 402). The surface brightness limit of a dataset is an important measure for extended structures (for example when you want to look at the outskirts of galaxies). In the next tutorial, we have thoroughly described the derivation of the surface brightness limit of a dataset. So we will just show the final result here, and encourage you to follow up with that tutorial after finishing this tutorial (see Section 2.3.4 [Image surface brightness limit], page 88).

By default, MakeCatalog will estimate the surface brightness limit of a given dataset, and put it in the keywords of the output (all keywords starting with SBL, which is short for surface brightness limit):

```
$ astfits cat/xdf-f160w.fits -h1 | grep SBL
```

As you see, the only one with a unit of mag/arcsec² is SBLMAG. It contains the surface brightness limit of the input dataset over SBLAREA arcsec² with SBLNSIG multiples of σ. In the current version of Gnuastro, SBLAREA=100 and SBLNSIG=3, so the surface brightness limit of this image is 32.66 mag/arcsec² (3σ, over 100 arcsec²). Therefore, if this default area and multiple of sigma are fine for you (these are the most commonly used values), you can simply read the image surface brightness limit from the catalogs produced by MakeCatalog with this command:

```
$ astfits cat/*.fits -h1 --keyvalue=SBLMAG
```

### 2.2.15 Working with catalogs (estimating colors)

In the previous step we generated catalogs of objects and clumps over our dataset (see Section 2.2.13 [Segmentation and making a catalog], page 52). The catalogs are available in the two extensions of the single FITS file. Let’s see the extensions and their basic properties with the Fits program:

```
$ astfits cat/xdf-f160w.fits # Extension information
```

Let’s inspect the table in each extension with Gnuastro’s Table program (see Section 5.3 [Table], page 230). We should have used -hOBJECTS and -hCLUMPS instead of -h1 and -h2 respectively. The numbers are just used here to convey that both names or numbers are possible, in the next commands, we’ll just use names.

```
$ asttable cat/xdf-f160w.fits -h1 --info # Objects catalog info.
$ asttable cat/xdf-f160w.fits -h1 # Objects catalog columns.
$ asttable cat/xdf-f160w.fits -h2 -i # Clumps catalog info.
$ asttable cat/xdf-f160w.fits -h2 # Clumps catalog columns.
```

As you see above, when given a specific table (file name and extension), Table will print the full contents of all the columns. To see the basic metadata about each column (for example name, units and comments), simply append a --info (or -i) to the command.

To print the contents of special column(s), just give the column number(s) (counting from 1) or the column name(s) (if they have one) to the --column (or -c) option. For

---

20 You can change these values with the --sfmagarea and --sfmagnsigma

21 MakeCatalog can also output plain text tables. However, in the plain text format you can only have one table per file. Therefore, if you also request measurements on clumps, two plain text tables will be created (suffixed with _o.txt and _c.txt).
example, if you just want the magnitude and signal-to-noise ratio of the clumps (in the clumps catalog), you can get it with any of the following commands

```
$ asttable cat/xdf-f160w.fits -hCLUMPS --column=5,6
$ asttable cat/xdf-f160w.fits -hCLUMPS -c5,SN
$ asttable cat/xdf-f160w.fits -hCLUMPS -c5 -c6
$ asttable cat/xdf-f160w.fits -hCLUMPS -cMAGNITUDE -cSN
```

Similar to HDUs, when the columns have names, always use the name: it is so common to mis-write numbers or forget the order later! Using column names instead of numbers has many advantages:

1. You don’t have to worry about the order of columns in the table.
2. It acts as a documentation in the script.
3. Column meta-data (including a name) aren’t just limited to FITS tables and can also be used in plain text tables, see Section 4.7.2 [Gnuastro text table format], page 185.

Table also has tools to limit the displayed rows. For example with the first command below only rows with a magnitude in the range of 29 to 30 will be shown. With the second command, you can further limit the displayed rows to rows with an S/N larger than 10 (a range between 10 to infinity). You can further sort the output rows, only show the top (or bottom) N rows and etc, for more see Section 5.3 [Table], page 230.

```
$ asttable cat/xdf-f160w.fits -hCLUMPS --range=MAGNITUDE,28:29
$ asttable cat/xdf-f160w.fits -hCLUMPS
 --range=MAGNITUDE,28:29 --range=SN,10:inf
```

Now that you are comfortable in viewing table columns and rows, let’s look into merging columns of multiple tables into one table (which is necessary for measuring the color of the clumps). Since `cat/xdf-f160w.fits` and `cat/xdf-f105w-on-f160w-lab.fits` have exactly the same number of rows and the rows correspond to the same clump, let’s merge them to have one table with magnitudes in both filters.

We can merge columns with the `--catcolumnfile` option like below. You give this option a file name (which is assumed to be a table that has the same number of rows as the main input), and all the table’s columns will be concatenated/appended to the main table. So please try it out with the commands below. We’ll first look at the metadata of the first table (only the `CLUMPS` extension). With the second command, we’ll concatenate the two tables and write them in `two-in-one.fits` and finally, we’ll check the new catalog’s metadata.

```
$ asttable cat/xdf-f160w.fits -i -hCLUMPS
$ asttable cat/xdf-f160w.fits -hCLUMPS
 --output=two-in-one.fits \
 --catcolumnfile=cat/xdf-f125w-on-f160w-lab.fits \
 --catcolumnhdu=CLUMPS
$ asttable two-in-one.fits -i
```

By comparing the two metadata, we see that both tables have the same number of rows. But what might have attracted your attention more, is that `two-in-one.fits` has double the number of columns (as expected, after all, you merged both tables into one file, and didn’t ask for any specific column). In fact you can concatenate any number of other tables in one command, for example:

```
$ asttable cat/xdf-f160w.fits -hCLUMPS --output=three-in-one.fits \
```

Chapter 2: Tutorials

As you see, to avoid confusion in column names, Table has intentionally appended a \(-1\) to the column names of the first concatenated table (so for example we have the original RA column, and another one called RA-1). Similarly a \(-2\) has been added for the columns of the second concatenated table.

However, this example clearly shows a problem with this full concatenation: some columns are identical (for example HOST_OBJ_ID and HOST_OBJ_ID-1), or not needed (for example RA-1 and DEC-1 which are not necessary here). In such cases, you can use \(--catcolumns\) to only concatenate certain columns, not the whole table. For example this command:

```bash
$ asttable cat/xdf-f160w.fits -hCLUMPS --output=two-in-one-2.fits \
  --catcolumnfile=cat/xdf-f125w-on-f160w-lab.fits \
  --catcolumnhdu=CLUMPS --catcolumns=MAGNITUDE
$ asttable two-in-one-2.fits -i
```

You see that we have now only appended the MAGNITUDE column of \(cat/xdf-f125w-on-f160w-lab.fits\). This is what we needed to be able to later subtract the magnitudes. Let’s go ahead and add the F105W magnitudes also with the command below. Note how we need to call \(--catcolumnhdu\) once for every table that should be appended, but we only call \(--catcolumn\) once (assuming all the tables that should be appended have this column).

```bash
$ asttable cat/xdf-f160w.fits -hCLUMPS --output=three-in-one-2.fits \
  --catcolumnfile=cat/xdf-f125w-on-f160w-lab.fits \
  --catcolumnhdu=CLUMPS --catcolumns=MAGNITUDE \
  --catcolumnfile=cat/xdf-f105w-on-f160w-lab.fits \
  --catcolumnhdu=CLUMPS --catcolumns=MAGNITUDE
$ asttable three-in-one-2.fits -i
```

But we aren’t finished yet! There is a very big problem: its not immediately clear which one of MAGNITUDE, MAGNITUDE-1 or MAGNITUDE-2 columns belong to which filter! Right now, you know this because you just ran this command. But in one hour, you’ll start doubting your self and will be forced to go through your command history, trying to figure out if you added F105W first, or F125W. You should never torture your future-self (or your colleagues) like this! So, let’s rename these confusing columns in the matched catalog.

Fortunately, with the \(--colmetadata\) option, you can correct the column metadata of the final table (just before it is written). It takes four values: 1) the original column name or number, 2) the new column name, 3) the column unit and 4) the column comments. Since the comments are usually human-friendly sentences and contain space characters, you should put them in double quotations like below. For example by adding three calls of this option to the previous command, we write the filter name in the magnitude column name and description.

```bash
$ asttable cat/xdf-f160w.fits -hCLUMPS --output=three-in-one-3.fits \
  --catcolumnfile=cat/xdf-f125w-on-f160w-lab.fits \
  --catcolumnfile=cat/xdf-f105w-on-f160w-lab.fits \
  --catcolumnhdu=CLUMPS --catcolumnhdu=CLUMPS \
  --catcolumn=RA-1 --colmetadata=RA-1=host_RA \"mag in F125w\" \
  --catcolumn=RA-2 --colmetadata=RA-2=host_RA \"mag in F105w\"
```
We now have all three magnitudes in one table and can start doing arithmetic on them (to estimate colors, which are just a subtraction of magnitudes). To use column arithmetic, simply call the column selection option (--column or -c), put the value in single quotations and start the value with arith (followed by a space) like the example below. Column arithmetic uses the same “reverse polish notation” as the Arithmetic program (see Section 6.2.1 [Reverse polish notation], page 272), with almost all the same operators (see Section 6.2.3 [Arithmetic operators], page 276), and some column-specific operators (that aren’t available for images). In column-arithmetic, you can identify columns by number (prefixed with a $) or name, for more see Section 5.3.1 [Column arithmetic], page 230.

So let’s estimate one color from three-in-one-3.fits using column arithmetic. All the commands below will produce the same output, try them each and focus on the differences. Note that column arithmetic can be mixed with other ways to choose output columns (the -c option).

$ asttable three-in-one-3.fits -ocolor-cat.fits \
-ocolor-cat.fits -c1,2,RA,DEC,'arith MAG-F125W MAG-F160W -'

This example again highlights the important point on using column names: if you don’t know the commands before, you have no way of making sense of the first command: what is in column 5 and 7? why not subtract columns 3 and 4 from each other? Do you see how cryptic the first one is? Then look at the last one: even if you have no idea how this table was created, you immediately understand the desired operation. When you have column names, please use them. If your table doesn’t have column names, give them names with the --colmetadata (described above) as you are creating them. But how about the metadata for the column you just created with column arithmetic? Have a look at the column metadata of the table produced above:

$ asttable color-cat.fits -i

The name of the column produced by arithmetic column is ARITH_1! This is natural: Arithmetic has no idea what the modified column is! You could have multiplied two columns, or done much more complex transformations with many columns. Metadata can’t be set automatically, your (the human) input is necessary. To add metadata, you can use --colmetadata like before:

$ asttable three-in-one-3.fits -ocolor-cat.fits -c1,2,RA,DEC \
-ocolor-cat.fits -c1,2,RA,DEC --column='arith MAG-F105W MAG-F160W -' \
-ocolor-cat.fits -c1,2,RA,DEC --column='arith MAG-F105W MAG-F160W -'

$ asttable color-cat.fits -i
We are now ready to make our final table. We want it to have the magnitudes in all three filters, as well as the three possible colors. Recall that by convention in astronomy colors are defined by subtracting the bluer magnitude from the redder magnitude. In this way a larger color value corresponds to a redder object. So from the three magnitudes, we can produce three colors (as shown below). Also, because this is the final table we are creating here and want to use it later, we’ll store it in cat/ and we’ll also give it a clear name and use the --range option to only print columns with a signal-to-noise ratio (SN column, from the F160W filter) above 5.

```bash
$ asttable cat/mags-with-color.fits -i
```

The table now has all the columns we need and it has the proper metadata to let us safely use it later (without frustrating over column orders!) or passing it to colleagues.

Let’s finish this section of the tutorial with a useful tip on modifying column metadata. Above, updating/changing column metadata was done with the --colmetadata in the same command that produced the newly created Table file. But in many situations, the table is already made and you just want to update the metadata of one column. In such cases using --colmetadata is over-kill (wasting CPU/RAM energy or time if the table is large) because it will load the full table data and metadata into memory, just change the metadata and write it back into a file.

In scenarios when the table’s data doesn’t need to be changed and you just want to set or update the metadata, it is much more efficient to use basic FITS keyword editing. For example, in the FITS standard, column names are stored in the TTYPE header keywords, so let’s have a look:

```bash
$ asttable two-in-one.fits -i
$ astfits two-in-one.fits -h1 | grep TTYPE
```

Changing/updating the column names is as easy as updating the values to these keywords. You don’t need to touch the actual data! With the command below, we’ll just update the MAGNITUDE and MAGNITUDE-1 columns (which are respectively stored in the TTYPE5 and TTYPE11 keywords) by modifying the keyword values and checking the effect by listing the column metadata again:

```bash
$ astfits two-in-one.fits -h1 \   --update=TTYPE5,MAG-F160W \   --update=TTYPE11,MAG-F125W
$ asttable two-in-one.fits -i
```
You can see that the column names have indeed been changed without touching any of the data. You can do the same for the column units or comments by modifying the keywords starting with \texttt{TUNIT} or \texttt{TCOMM}.

Generally, Gnuastro’s table is a very useful program in data analysis and what you have seen so far is just the tip of the iceberg. But to avoid making the tutorial even longer, we’ll stop reviewing the features here, for more, please see Section 5.3 [Table], page 230. Before continuing, let’s just delete all the temporary FITS tables we placed in the top project directory:

\begin{verbatim}
rm *.fits
\end{verbatim}

\subsection*{2.2.16 Column statistics (color-magnitude diagram)}

In Section 2.2.15 [Working with catalogs (estimating colors)], page 58, we created a single catalog containing the magnitudes of our desired clumps in all three filters, and their colors. To start with, let’s inspect the distribution of three colors with the Statistics program.

\begin{verbatim}
$ aststatistics cat/mags-with-color.fits -cF105W-F125W
$ aststatistics cat/mags-with-color.fits -cF105W-F160W
$ aststatistics cat/mags-with-color.fits -cF125W-F160W
\end{verbatim}

This tiny and cute ASCII histogram (and the general information printed above it) gives you a crude (but very useful and fast) feeling on the distribution. You can later use Gnuastro’s Statistics program with the \texttt{--histogram} option to build a much more fine-grained histogram as a table to feed into your favorite plotting program for a much more accurate/appealing plot (for example with PGFPlots in \LaTeX). If you just want a specific measure, for example the mean, median and standard deviation, you can ask for them specifically, like below:

\begin{verbatim}
$ aststatistics cat/mags-with-color.fits -cF105W-F160W \
   --mean --median --std
\end{verbatim}

The basic statistics we measured above were just on one column. In many scenarios this is fine, but things get much more exciting if you look at the correlation of two columns with each other. For example, let’s create the color-magnitude diagram for our measured targets.

In many papers, the color-magnitude diagram is usually plotted as a scatter plot. However, scatter plots have a major limitation when there are a lot of points and they cluster together in one region of the plot: the possible correlation in that dense region is lost (because the points fall over each other). In such cases, its much better to use a 2D histogram. In a 2D histogram, the full range in both columns is divided into discrete 2D bins (or pixels!) and we count how many objects fall in that 2D bin.

Since a 2D histogram is a pixelated space, we can simply save it as a FITS image and view it in a FITS viewer. Let’s do this in the command below. As is common with color-magnitude plots, we’ll put the redder magnitude on the horizontal axis and the color on the vertical axis. We’ll set both dimensions to have 100 bins (with \texttt{--numbins} for the horizontal and \texttt{--numbins2} for the vertical). Also, to avoid strong outliers in any of the dimensions, we’ll manually set the range of each dimension with the \texttt{--greaterequal}, \texttt{--greaterequal2}, \texttt{--lessthan} and \texttt{--lessthan2} options.

\begin{verbatim}
$ aststatistics cat/mags-with-color.fits -cMAG-F160W,F105W-F160W \
   --histogram2d=image --manualbinrange \
\end{verbatim}
You can now open this FITS file as a normal FITS image, for example with the command below. Try hovering/zooming over the pixels: not only will you see the number of objects in the UVUDF catalog that fall in each bin/pixel, but you also see the $F_{160W}$ magnitude and color of that pixel also (in the same place you usually see RA and Dec when hovering over an astronomical image).

```bash
$ ds9 cmd.fits -cmap sls -zoom to fit
```

Having a 2D histogram as a FITS image with WCS has many great advantages. For example, just like FITS images of the night sky, you can “match” many 2D histograms that were created independently. You can add two histograms with each other, or you can use advanced features of FITS viewers to find structure in the correlation of your columns.

With the first command below, you can activate the grid feature of DS9 to actually see the coordinate grid, as well as values on each line. With the second command, DS9 will even read the labels of the axes and use them to generate an almost publication-ready plot.

```bash
$ ds9 cmd.fits -cmap sls -zoom to fit -grid yes
$ ds9 cmd.fits -cmap sls -zoom to fit -grid yes -grid type publication
```

If you are happy with the grid and coloring and etc, you can also use ds9 to save this as a JPEG image to directly use in your documents/slides with these extra DS9 options (DS9 will write the image to `cmd-2d.jpeg` and quit immediately afterwards):

```bash
$ ds9 cmd.fits -cmap sls -zoom 4 -grid yes -grid type publication \  -saveimage cmd-2d.jpeg -quit
```

This is good for a fast progress update. But for your paper or more official report, you want to show something with higher quality. For that, you can use the PGFPLOTS package in \LaTeX to add axises in the same font as your text, sharp grids and many other elegant/powerful features (like over-plotting interesting points, lines and etc). But to load the 2D histogram into PGFPLOTs first you need to convert the FITS image into a more standard format, for example PDF. We’ll use Gnuastro’s Section 5.2 [ConvertType], page 211, for this, and use the `sls-inverse` color map (which will map the pixels with a value of zero to white):

```bash
$ astconvertt cmd.fits --colormap=sls-inverse --borderwidth=0 -ocmd.pdf
```

Below you can see a minimally working example of how to add axis numbers, labels and a grid to the PDF generated above. First, let’s create a new `report` directory to keep the \LaTeX outputs, then put the minimal report’s source in a file called `report.tex`. Notice the $\text{xmin}$, $\text{xmax}$, $\text{ymin}$, $\text{ymax}$ values and how they are the same as the range specified above.

```bash
$ mkdir report
$ mv cmd.pdf report/
$ cat report/report.tex
\documentclass{article}
\usepackage{pgfplots}
\dimendef\prevdepth=0
\begin{document}
```
You can write all you want here...

\begin{tikzpicture}
\begin{axis}[
  enlargelimits=false,
  grid,
  axis on top,
  width=\linewidth,
  height=\linewidth,
  xlabel={Magnitude (F160W)},
  ylabel={Color (F105W-F160W)}
]
\addplot graphics[xmin=22, xmax=30, ymin=-1, ymax=3] {cmd.pdf};
\end{axis}
\end{tikzpicture}

Run this command to build your PDF (assuming you have \LaTeX and PGFPlots).

$ cd \text{report}$

$ pdflatex \text{report.tex}$

Open the newly created \text{report.pdf} and enjoy the exquisite quality. The improved quality, blending in with the text, vector-graphics resolution and other features make this plot pleasing to the eye, and let your readers focus on the main point of your scientific argument. PGFPlots can also built the PDF of the plot separately from the rest of the paper/report, see Section 7.1.2.1 \[2D histogram as a table\], page 339, for the necessary changes in the preamble.

We won’t go much deeper into the Statistics program here, but there is so much more you can do with it. After finishing the tutorial, see Section 7.1 \[Statistics\], page 337.

\section*{2.2.17 Aperture photometry}

The colors we calculated in Section 2.2.15 \[Working with catalogs (estimating colors)\], page 58, used a different segmentation map for each object. This might not satisfy some science cases that need the flux within a fixed area/aperture. Fortunately Gnuastro’s modular programs make it very easy do this type of measurement (photometry). To do this, we can ignore the labeled images of NoiseChisel of Segment, we can just built our own labeled image! That labeled image can then be given to \text{MakeCatalog}

To generate the apertures catalog we’ll use Gnuastro’s \text{MakeProfiles} (see Section 8.1 \[MakeProfiles\], page 440). But first we need a list of positions (aperture photometry needs a-priori knowledge of your target positions). So we’ll first read the clump positions from the F160W catalog, then use AWK to set the other parameters of each profile to be a fixed circle of radius 5 pixels (recall that we want all apertures to have an identical size/area in this scenario).

$ rm *.fits *.txt$

$ asttable \text{cat/xdf-f160w.fits -hCLUMPS -cRA,DEC} \$

\vspace{1em}

\$ \text{awk '!/\#/{print NR, $1, $2, 5, 5, 0, 0, 1, NR, 1}'} \$

\vspace{1em}

\$ \text{> apertures.txt} \$
We can now feed this catalog into MakeProfiles using the command below to build the apertures over the image. The most important option for this particular job is --mforflatpix, it tells MakeProfiles that the values in the magnitude column should be used for each pixel of a flat profile. Without it, MakeProfiles would build the profiles such that the sum of the pixels of each profile would have a magnitude (in log-scale) of the value given in that column (what you would expect when simulating a galaxy for example). See Section 8.1.4 [Invoking MakeProfiles], page 447, for details on the options.

```bash
$ astmkprof apertures.txt --background=flat-ir/xdf-f160w.fits \
  --clearcanvas --replace --type=int16 --mforflatpix \
  --mode=wcs --output=apertures.fits
```

Open apertures.fits with a FITS image viewer (like SAO DS9) and look around at the circles placed over the targets. Also open the input image and Segment’s clumps image and compare them with the positions of these circles. Where the apertures overlap, you will notice that one label has replaced the other (because of the --replace option). In the future, MakeCatalog will be able to work with overlapping labels, but currently it doesn’t. If you are interested, please join us in completing Gnuastro with added improvements like this (see task 14750).22

We can now feed the apertures.fits labeled image into MakeCatalog instead of Segment’s output as shown below. In comparison with the previous MakeCatalog call, you will notice that there is no more --clumpscat option, since there is no more separate “clump” image now, each aperture is treated as a separate “object”.

```bash
$ astmkcatalog apertures.fits -h1 --zeropoint=26.27 \
  --valuesfile=nc/xdf-f105w.fits \
  --ids --ra --dec --magnitude --sn \
  --output=cat/xdf-f105w-aper.fits
```

This catalog has the same number of rows as the catalog produced from clumps in Section 2.2.15 [Working with catalogs (estimating colors)], page 58. Therefore similar to how we found colors, you can compare the aperture and clump magnitudes for example.

You can also change the filter name and zero point magnitudes and run this command again to have the fixed aperture magnitude in the F160W filter and measure colors on apertures.

### 2.2.18 Matching catalogs

In the example above, we had the luxury to generate the catalogs ourselves, and where thus able to generate them in a way that the rows match. But this isn’t generally the case. In many situations, you need to use catalogs from many different telescopes, or catalogs with high-level calculations that you can’t simply regenerate with the same pixels without spending a lot of time or using heavy computation. In such cases, when each catalog has the coordinates of its own objects, you can use the coordinates to match the rows with Gnuastro’s Match program (see Section 7.5 [Match], page 429).

As the name suggests, Gnuastro’s Match program will match rows based on distance (or aperture in 2D) in one, two, or three columns. For this tutorial, let’s try matching the two

catalogs that weren’t created from the same labeled images, recall how each has a different number of rows:

```bash
$ asttable cat/xdf-f105w.fits -hCLUMPS -i
$ asttable cat/xdf-f160w.fits -hCLUMPS -i
```

You give Match two catalogs (from the two different filters we derived above) as argument, and the HDUs containing them (if they are FITS files) with the `--hdu` and `--hdu2` options. The `--ccol1` and `--ccol2` options specify the coordinate-columns which should be matched with which in the two catalogs. With `--aperture` you specify the acceptable error (radius in 2D), in the same units as the columns.

```bash
$ astmatch cat/xdf-f160w.fits cat/xdf-f105w.fits
   --hdu=CLUMPS --hdu2=CLUMPS
   --ccol1=RA,DEC --ccol2=RA,DEC
   --aperture=0.5/3600 
   --output=matched.fits
$ astfits matched.fits
```

From the second command, you see that the output has two extensions and that both have the same number of rows. The rows in each extension are the matched rows of the respective input table: those in the first HDU come from the first input and those in the second HDU come from the second. However, their order may be different from the input tables because the rows match: the first row in the first HDU matches with the first row in the second HDU, and etc. You can also see which objects didn’t match with the `--notmatched`, like below. Note how each extension of now has a different number of rows.

```bash
$ astmatch cat/xdf-f160w.fits cat/xdf-f105w.fits
   --hdu=CLUMPS --hdu2=CLUMPS
   --ccol1=RA,DEC --ccol2=RA,DEC
   --aperture=0.5/3600 
   --output=not-matched.fits --notmatched
$ astfits not-matched.fits
```

The `--outcols` of Match is a very convenient feature: you can use it to specify which columns from the two catalogs you want in the output (merge two input catalogs into one). If the first character is an ‘a’, the respective matched column (number or name, similar to Table above) in the first catalog will be written in the output table. When the first character is a ‘b’, the respective column from the second catalog will be written in the output. Also, if the first character is followed by _all, then all the columns from the respective catalog will be put in the output.

```bash
$ astmatch cat/xdf-f160w.fits cat/xdf-f105w.fits
   --hdu=CLUMPS --hdu2=CLUMPS
   --ccol1=RA,DEC --ccol2=RA,DEC
   --aperture=0.35/3600 
   --outcols=a_all,bMAGNITUDE,bSN 
   --output=matched.fits
$ astfits matched.fits
```
2.2.19 Finding reddest clumps and visual inspection

As a final step, let’s go back to the original clumps-based color measurement we generated in Section 2.2.15 [Working with catalogs (estimating colors)], page 58. We’ll find the objects with the strongest color and make a cutout to inspect them visually and finally, we’ll see how they are located on the image. With the command below, we’ll select the reddest objects (those with a color larger than 1.5):

```
$ asttable cat/mags-with-color.fits --range=F105W-F160W,1.5,inf
```

You can see how many they are by piping it to `wc -l`:

```
$ asttable cat/mags-with-color.fits --range=F105W-F160W,1.5,inf | wc -l
```

Let’s crop the F160W image around each of these objects, but we first need a unique identifier for them. We’ll define this identifier using the object and clump labels (with an underscore between them) and feed the output of the command above to AWK to generate a catalog. Note that since we are making a plain text table, we’ll define the necessary (for the string-type first column) metadata manually (see Section 4.7.2 [Gnuastro text table format], page 185).

```
$ echo "# Column 1: ID [name, str10] Object ID" > reddest.txt
$ asttable cat/mags-with-color.fits --range=F105W-F160W,1.5,inf | awk '{printf("%d_%-10d %f %f\n", $1, $2, $3, $4)}' >> reddest.txt
```

We can now feed `reddest.txt` into Gnuastro’s Crop program to see what these objects look like. To keep things clean, we’ll make a directory called `crop-red` and ask Crop to save the crops in this directory. We’ll also add a `-f160w.fits` suffix to the crops (to remind us which filter they came from). The width of the crops will be 15 arc-seconds (or 15/3600 degrees, which is the units of the WCS).

```
$ mkdir crop-red
$ astcrop flat-ir/xdf-f160w.fits --mode=wcs --namecol=ID --catalog=reddest.txt --width=15/3600,15/3600 --suffix=-f160w.fits --output=crop-red
```

You can see all the cropped FITS files in the `crop-red` directory. Like the MakeProfiles command in Section 2.2.17 [Aperture photometry], page 65, you might notice that the crops aren’t made in order. This is because each crop is independent of the rest, therefore crops are done in parallel, and parallel operations are asynchronous. In the command above, you can change `f160w` to `f105w` to make the crops in both filters.

To view the crops more easily (not having to open ds9 for each image), you can convert the FITS crops into the JPEG format with a shell loop like below.

```
$ cd crop-red
$ for f in *.fits; do
    astconvertt $f --fluxlow=-0.001 --fluxhigh=0.005 --invert -ojpg;
    done
$ cd ..
$ ls crop-red/
```

You can now use your general graphic user interface image viewer to flip through the images more easily, or import them into your papers/reports.
The `for` loop above to convert the images will do the job in series: each file is converted only after the previous one is complete. If you have GNU Parallel (https://www.gnu.org/s/parallel), you can greatly speed up this conversion. GNU Parallel will run the separate commands simultaneously on different CPU threads in parallel. For more information on efficiently using your threads, see Section 4.4 [Multi-threaded operations], page 175. Here is a replacement for the shell `for` loop above using GNU Parallel.

```bash
$ cd crop-red
$ parallel astconvertt --fluxlow=-0.001 --fluxhigh=0.005 --invert -ojpg ::: *.fits
$ cd ..
```

Did you notice how much faster this one was? When possible, it’s always very helpful to do your analysis in parallel. But the problem is that many operations are not as simple as this. For such cases, you can use Make (https://en.wikipedia.org/wiki/Make_(software)) which will greatly help designing workflows. But that is beyond the topic here.

As the final action, let's see how these objects are positioned over the dataset. DS9 has the “Region”s concept for this purpose. You just have to convert your catalog into a “region file” to feed into DS9. To do that, you can use AWK again as shown below.

```bash
$ awk 'BEGIN{print "# Region file format: DS9 version 4.1"; print "global color=green width=2"; print "fk5";}
!/^#/{{printf "circle(%.2f,%.2f,1") # text={%.2f,.2f,%.2f,%.2f}n",$2,$3,$1;}}'
reddest.txt > reddest.reg
```

This region file can be loaded into DS9 with its `-regions` option to display over any image (that has world coordinate system). In the example below, we’ll open Segment’s output and load the regions over all the extensions (to see the image and the respective clump):

```bash
$ ds9 -mecube seg/xdf-f160w.fits -zscale -zoom to fit
 -regions load all reddest.reg
```

### 2.2.20 Writing scripts to automate the steps

In the previous sub-sections, we went through a series of steps like downloading the necessary datasets (in Section 2.2.3 [Setup and data download], page 31), detecting the objects in the image, and finally selecting a particular subset of them to inspect visually (in Section 2.2.19 [Finding reddest clumps and visual inspection], page 68). To benefit most effectively from this subsection, please go through the previous sub-sections, and if you haven’t actually done them, we recommended to do/run them before continuing here.

Each sub-section/step of the sub-sections above involved several commands on the command-line. Therefore, if you want to reproduce the previous results (for example to only change one part, and see its effect), you’ll have to go through all the sections above and read through them again. If you done the commands recently, you may also have them in the history of your shell (command-line environment). You can see many of your previous commands on the shell (even if you have closed the terminal) with the `history` command, like this:

```bash
$ history
```
Try it in your terminal to see for yourself. By default in GNU Bash, it shows the last 500 commands. You can also save this “history” of previous commands to a file using shell redirection (to have it after your next 500 commands), with this command

\$ history > my-previous-commands.txt

This is a good way to temporarily keep track of every single command you ran. But in the middle of all the useful commands, you will have many extra commands, like tests that you did before/after the good output of a step (that you decided to continue working on), or an unrelated job you had to do in the middle of this project. Because of these impurities, after a few days (that you have forgot the context: tests you didn’t end-up using, or unrelated jobs) reading this full history will be very frustrating.

Keeping the final commands that were used in each step of an analysis is a common problem for anyone who is doing something serious with the computer. But simply keeping the most important commands in a text file is not enough, the small steps in the middle (like making a directory to keep the outputs of one step) are also important. In other words, the only way you can be sure that you are under control of your processing (and actually understand how you produced your final result) is to run the commands automatically.

Fortunately, typing commands interactively with your fingers isn’t the only way to operate the shell. The shell can also take its orders/commands from a plain-text file, which is called a script. When given a script, the shell will read it line-by-line as if you have actually typed it manually.

Let’s continue with an example: try typing the commands below in your shell. With these commands we are making a text file (a.txt) containing a simple 3 × 3 matrix, converting it to a FITS image and computing its basic statistics. After the first three commands open a.txt with a text editor to actually see the values we wrote in it, and after the fourth, open the FITS file to see the matrix as an image. a.txt is created through the shell’s redirection feature: ‘>' overwrites the existing contents of a file, and ‘>>' appends the new contents after the old contents.

\$ echo "1 1 1" > a.txt
\$ echo "1 2 1" >> a.txt
\$ echo "1 1 1" >> a.txt
\$ astconvertt a.txt --output=a.fits
\$ aststatistics a.fits

To automate these series of commands, you should put them in a text file. But that text file must have two special features: 1) It should tell the shell what program should interpret the script. 2) The operating system should know that the file can be directly executed.

For the first, Unix-like operating systems define the shebang concept (also known as sha-bang or hashbang). In the shebang convention, the first two characters of a file should be ‘#!’. When confronted with these characters, the script will be interpreted with the program that follows them. In this case, we want to write a shell script and the most common shell program is GNU Bash which is installed in /bin/bash. So the first line of your script should be ‘#!/bin/bash’\(^{23}\).

---

\(^{23}\) When the script is to be run by the same shell that is calling it (like this script), the shebang is optional. But it is still recommended, because it ensures that even if the user isn’t using GNU Bash, the script will be run in GNU Bash: given the differences between various shells, writing truly portable shell scripts, that can be run by many shell programs/implementations, isn’t easy (sometimes not possible!).
It may happen (rarely) that GNU Bash is in another location on your system. In other cases, you may prefer to use a non-standard version of Bash installed in another location (that has higher priority in your PATH, see Section 3.3.1.2 [Installation directory], page 139). In such cases, you can use the ‘#!/usr/bin/env bash’ shebang instead. Through the env program, this shebang will look in your PATH and use the first bash it finds to run your script. But for simplicity in the rest of the tutorial, we’ll continue with the ‘#!/bin/bash’ shebang.

Using your favorite text editor, make a new empty file, let’s call it my-first-script.sh. Write the GNU Bash shebang (above) as its first line. After the shebang, copy the series of commands we ran above. Just note that the ‘$’ sign at the start of every line above is the prompt of the interactive shell (you never actually typed it, remember?). Therefore, commands in a shell script should not start with a ‘$’. Once you add the commands, close the text editor and run the cat command to confirm its contents. It should look like the example below. Recall that you should only type the line that starts with a ‘$’, the lines without a ‘$’, are printed automatically on the command-line (they are the contents of your script).

```
$ cat my-first-script.sh
#!/bin/bash
echo "1 1 1" > a.txt
echo "1 2 1" >> a.txt
echo "1 1 1" >> a.txt
astconvertt a.txt --output=a.fits
aststatistics a.fits
```

The script contents are now ready, but to run it, you should activate the script file’s executable flag. In Unix-like operating systems, every file has three types of flags: read (or r), write (or w) and execute (or x). To toggle a file’s flags, you should use the chmod (for “change mode”) command. To activate a flag, you put a ‘+’ before the flag character (for example +x). To deactivate it, you put a ‘-’ (for example -x). In this case, you want to activate the script’s executable flag, so you should run

```
$ chmod +x my-first-script.sh
```

Your script is now ready to run/execute the series of commands. To run it, you should call it while specifying its location in the file system. Since you are currently in the same directory as the script, its easiest to use relative addressing like below (where ‘./’ means the current directory). But before running your script, first delete the two a.txt and a.fits files that were created when you interactively ran the commands.

```
$ rm a.txt a.fits
$ ls
$ ./my-first-script.sh
$ ls
```

The script immediately prints the statistics while doing all the previous steps in the background. With the last ls, you see that it automatically re-built the a.txt and a.fits files, open them and have a look at their contents.

An extremely useful feature of shell scripts is that the shell will ignore anything after a ‘#’ character. You can thus add descriptions/comments to the commands and make them
much more useful for the future. For example, after adding comments, your script might look like this:

```bash
#!/bin/bash

# This script is my first attempt at learning to write shell scripts.
# As a simple series of commands, I am just building a small FITS image, and calculating its basic statistics.

# Write the matrix into a file.
echo "1 1 1" > a.txt
echo "1 2 1" >> a.txt
echo "1 1 1" >> a.txt

# Convert the matrix to a FITS image.
astconvertt a.txt --output=a.fits

# Calculate the statistics of the FITS image.
aststatistics a.fits
```

Isn’t this much more easier to read now? Comments help to provide human-friendly context to the raw commands. At the time you make a script, comments may seem like an extra effort and slow you down. But in one year, you will forget almost everything about your script and you will appreciate the effort so much! Think of the comments as an email to your future-self and always put a well-written description of the context/purpose (most importantly, things that aren’t directly clear by reading the commands) in your scripts.

The example above was very basic and mostly redundant series of commands, to show the basic concepts behind scripts. You can put any (arbitrarily long and complex) series of commands in a script by following the two rules: 1) add a shebang, and 2) enable the executable flag. In fact, as you continue your own research projects, you will find that any time you are dealing with more than two or three commands, keeping them in a script (and modifying that script, and running it) is much more easier, and future-proof, then typing the commands directly on the command-line and relying on things like `history`.

Here are some tips that will come in handy when you are writing your scripts:

As a more realistic example, let’s have a look at a script that will do the steps of Section 2.2.3 [Setup and data download], page 31, and Section 2.2.4 [Dataset inspection and cropping], page 32. In particular note how often we are using variables to avoid repeating fixed strings of characters (usually file/directory names). This greatly helps in scaling up your project, and avoiding hard-to-find bugs that are caused by typos in those fixed strings.

```bash
#!/bin/bash

# Download the input datasets
# -----------------------------

# The default file names have this format (where ‘FILTER’ differs for
# each filter):
#   hlsp_xdf_hst_wfc3ir-60mas_hudf_FILTER_v1_sci.fits
# To make the script easier to read, a prefix and suffix variable are
# used to sandwich the filter name into one short line.
downloaddir=download
xdfsuffix=_v1_sci.fits
xdfprefix=hlsp_xdf_hst_wfc3ir-60mas_hudf_
xdfurl=http://archive.stsci.edu/pub/hlsp/xdf

# The file name and full URLs of the input data.
f105w_in=$xdfprefix"f105w"$xdfsuffix
f160w_in=$xdfprefix"f160w"$xdfsuffix
f105w_full=$xdfurl/$f105w_in
f160w_full=$xdfurl/$f160w_in

# Go into the download directory and download the images there,
# then come back up to the top running directory.
mkdir $downloaddir
cd $downloaddir
wget $f105w_full
wget $f160w_full
cd ..

# Only work on the deep region
# -----------------------------
#
# To help in readability, each vertice of the deep/flat field is stored
# as a separate variable. They are then merged into one variable to
# define the polygon.
flatdir=flat-ir
vertice1="53.187414,-27.779152"
vertice2="53.159507,-27.759633"
vertice3="53.134517,-27.787144"
vertice4="53.161906,-27.807208"
f105w_flat=$flatdir/xdf-f105w.fits
f160w_flat=$flatdir/xdf-f160w.fits
deep_polygon="$vertice1:$vertice2:$vertice3:$vertice4"

mkdir $flatdir
astcrop --mode=wcs -h0 --output=$f105w_flat \       --polygon=$deep_polygon $downloaddir/$f105w_in
astcrop --mode=wcs -h0 --output=$f160w_flat \      --polygon=$deep_polygon $downloaddir/$f160w_in

The first thing you may notice is that even if you already have the downloaded input
images, this script will always try to re-download them. Also, if you re-run the script, you
will notice that `mkdir` prints an error message that the download directory already exists. Therefore, the script above isn’t too useful and some modifications are necessary to make it more generally useful. Here are some general tips that are often very useful when writing scripts:

**Stop script if a command crashes**

By default, if a command in a script crashes (aborts and fails to do what it was meant to do), the script will continue onto the next command. In GNU Bash, you can tell the shell to stop a script in the case of a crash by adding this line at the start of your script:

```
set -e
```

**Check if a file/directory exists to avoid re-creating it**

Conditionals are a very useful feature in scripts. One common conditional is to check if a file exists or not. Assuming the file’s name is `FILENAME`, you can check its existence (to avoid re-doing the commands that build it) like this:

```
if [ -f FILENAME ]; then
    echo "FILENAME exists"
else
    # Some commands to generate the file
    echo "done" > FILENAME
fi
```

To check the existence of a directory instead of a file, use `-d` instead of `-f`. To negate a conditional, use `!’` and note that conditionals can be written in one line also (useful for when its short).

One common scenario that you’ll need to check the existence of directories is when you are making them: the default `mkdir` command will crash if the desired directory already exists. On some systems (including GNU/Linux distributions), `mkdir` has options to deal with such cases. But if you want your script to be portable, its best to check yourself like below:

```
if ! [ -d DIRNAME ]; then mkdir DIRNAME; fi
```

Taking these tips into consideration, we can write a better version of the script above that includes checks on every step to avoid repeating steps/commands. Please compare this script with the previous one carefully to spot the differences. These are very important points that you will definitely encounter during your own research, and knowing them can greatly help your productivity, so pay close attention (even in the comments).

```
$ cat gnuastro-tutorial-2.sh
#!/bin/bash
set -e

# Download the input datasets
# ---------------------------
#
# The default file names have this format (where ‘FILTER’ differs for # each filter):
```
# hlsp_xdf_hst_wfc3ir-60mas_hudf_FILTER_v1_sci.fits
# To make the script easier to read, a prefix and suffix variable are
# used to sandwich the filter name into one short line.
downloaddir=download
xdfsuffix=_v1_sci.fits
xdfprefix=hlsp_xdf_hst_wfc3ir-60mas_hudf_
xdfurl=http://archive.stsci.edu/pub/hlsp/xdf

# The file name and full URLs of the input data.
f105w_in=$xdfprefix"f105w"$xdfsuffix
f160w_in=$xdfprefix"f160w"$xdfsuffix
f105w_full=$xdfurl/$f105w_in
f160w_full=$xdfurl/$f160w_in

# Go into the download directory and download the images there,
# then back up to the top running directory.
if ! [ -d $downloaddir ]; then mkdir $downloaddir; fi
cd $downloaddir
if ! [ -f $f105w_in ]; then wget $f105w_full; fi
if ! [ -f $f160w_in ]; then wget $f160w_full; fi
cd ..

# Only work on the deep region
# -----------------------------

# To help in readability, each vertice of the deep/flat field is stored
# as a separate variable. They are then merged into one variable to
# define the polygon.
flatdir=flat-ir
vertice1=53.187414,-27.779152
vertice2=53.159507,-27.759633
vertice3=53.134517,-27.787144
vertice4=53.161906,-27.807208
f105w_flat=$flatdir/xdf-f105w.fits
f160w_flat=$flatdir/xdf-f160w.fits
deep_polygon="$vertice1:$vertice2:$vertice3:$vertice4"

if ! [ -d $flatdir ]; then mkdir $flatdir; fi
if ! [ -f $f105w_flat ]; then
    astcrop --mode=wcs -h0 --output=$f105w_flat \
        --polygon=$deep_polygon $downloaddir/$f105w_in
fi
if ! [ -f $f160w_flat ]; then
    astcrop --mode=wcs -h0 --output=$f160w_flat \
        --polygon=$deep_polygon $downloaddir/$f160w_in
fi
2.2.21 Citing and acknowledging Gnuastro

In conclusion, we hope this extended tutorial has been a good starting point to help in your exciting research. If this book or any of the programs in Gnuastro have been useful for your research, please cite the respective papers, and acknowledge the funding agencies that made all of this possible. Without citations, we won’t be able to secure future funding to continue working on Gnuastro or improving it, so please take software citation seriously (for all the scientific software you use, not just Gnuastro).

To help you in this aspect is well, all Gnuastro programs have a `--cite` option to facilitate the citation and acknowledgment. Just note that it may be necessary to cite additional papers for different programs, so please try it out on all the programs that you used, for example:

```
$ astmkcatalog --cite
$ astnoisechisel --cite
```

2.3 Detecting large extended targets

The outer wings of large and extended objects can sink into the noise very gradually and can have a large variety of shapes (for example due to tidal interactions). Therefore separating the outer boundaries of the galaxies from the noise can be particularly tricky. Besides causing an under-estimation in the total estimated brightness of the target, failure to detect such faint wings will also cause a bias in the noise measurements, thereby hampering the accuracy of any measurement on the dataset. Therefore even if they don’t constitute a significant fraction of the target’s light, or aren’t your primary target, these regions must not be ignored. In this tutorial, we’ll walk you through the strategy of detecting such targets using Section 7.2 [NoiseChisel], page 362.

Don’t start with this tutorial: If you haven’t already completed Section 2.2 [General program usage tutorial], page 29, we strongly recommend going through that tutorial before starting this one. Basic features like access to this book on the command-line, the configuration files of Gnuastro’s programs, benefiting from the modular nature of the programs, viewing multi-extension FITS files, or using NoiseChisel’s outputs are discussed in more detail there.

We’ll try to detect the faint tidal wings of the beautiful M51 group\(^{24}\) in this tutorial. We’ll use a dataset/image from the public Sloan Digital Sky Survey (http://www.sdss.org/), or SDSS. Due to its more peculiar low surface brightness structure/features, we’ll focus on the dwarf companion galaxy of the group (or NGC 5195).

2.3.1 Downloading and validating input data

To get the image, you can use SDSS’s Simple field search (https://dr12.sdss.org/fields) tool. As long as it is covered by the SDSS, you can find an image containing your desired target either by providing a standard name (if it has one), or its coordinates. To access the dataset we will use here, write NGC5195 in the “Object Name” field and press “Submit” button.

\(^{24}\) https://en.wikipedia.org/wiki/M51_Group
Type the example commands: Try to type the example commands on your terminal and use the history feature of your command-line (by pressing the “up” button to retrieve previous commands). Don’t simply copy and paste the commands shown here. This will help simulate future situations when you are processing your own datasets.

You can see the list of available filters under the color image. For this demonstration, we’ll use the r-band filter image. By clicking on the “r-band FITS” link, you can download the image. Alternatively, you can just run the following command to download it with GNU Wget. To keep things clean, let’s also put it in a directory called ngc5195. With the -0 option, we are asking Wget to save the downloaded file with a more manageable name: r.fits.bz2 (this is an r-band image of NGC 5195, which was the directory name).

```bash
$ mkdir ngc5195
$ cd ngc5195
$ topurl=https://dr12.sdss.org/sas/dr12/boss/photoObj/frames
$ wget $topurl/301/3716/6/frame-r-003716-6-0117.fits.bz2 -Or.fits.bz2
```

When you want to reproduce a previous result (a known analysis, on a known dataset, to get a known result: like the case here!) it is important to verify that the file is correct: that the input file hasn’t changed (on the remote server, or in your own archive), or there was no downloading problem. Otherwise, if the data have changed in your server/archive, and you use the same script, you will get a different result, causing a lot of confusion!

One good way to verify the contents of a file is to store its Checksum in your analysis script and check it before any other operation. The Checksum algorithms look into the contents of a file and calculate a fixed-length string from them. If any change (even in a bit or byte) is made within the file, the resulting string will change, for more see Wikipedia (https://en.wikipedia.org/wiki/Checksum). There are many common algorithms, but a simple one is the SHA-1 algorithm (https://en.wikipedia.org/wiki/SHA-1) (Secure Hash Algorithm 1) that you can calculate easily with the command below (the second line is the output, and the checksum is the first/long string: it is independent of the file name)

```bash
$ sha1sum r.fits.bz2
5fb06a572c6107c72cbc5eb8a9329f536c7e7f65  r.fits.bz2
```

If the checksum on your computer is different from this, either the file has been incorrectly downloaded (most probable), or it has changed on SDSS servers (very unlikely). To get a better feeling of checksums open your favorite text editor and make a test file by writing something in it. Save it and calculate the text file’s SHA-1 checksum with sha1sum. Try renaming that file, and you’ll see the checksum hasn’t changed (checksums only look into the contents, not the name/location of the file). Then open the file with your text editor again, make a change and re-calculate its checksum, you’ll see the checksum string has changed.

It’s always good to keep this short checksum string with your project’s scripts and validate your input data before using them. You can do this with a shell conditional like this:

---

25 To make the command easier to view on screen or in a page, we have defined the top URL of the image as the topurl shell variable. You can just replace the value of this variable with $topurl in the wget command.

26 If your checksum is different, try uncompressing the file with the bunzip2 command after this, and open the resulting FITS file. If it opens and you see the image of M51 and NGC5195, then there was no download problem, and the file has indeed changed on the SDSS servers! In this case, please contact us at bug-gnuastro@gnu.org.
Now that we know you have the same data that we wrote this tutorial with, let’s continue. The SDSS server keeps the files in a Bzip2 compressed file format (that have a .bz2 suffix). So we’ll first decompress it with the following command to use it as a normal FITS file. By convention, compression programs delete the original file (compressed when uncompressing, or uncompressed when compressing). To keep the original file, you can use the --keep or -k option which is available in most compression programs for this job. Here, we don’t need the compressed file any more, so we’ll just let bunzip delete it for us and keep the directory clean.

```bash
$ bunzip2 r.fits.bz2
```

### 2.3.2 NoiseChisel optimization

In Section 2.3 [Detecting large extended targets], page 76, we downloaded the single exposure SDSS image. Let’s see how NoiseChisel operates on it with its default parameters:

```bash
$ astnoisechisel r.fits -h0
```

As described in Section 2.2.10 [NoiseChisel and Multiextension FITS files], page 44, NoiseChisel’s default output is a multi-extension FITS file. Open the output r_detected.fits file and have a look at the extensions, the 0-th extension is only meta-data and contains NoiseChisel’s configuration parameters. The rest are the Sky-subtracted input, the detection map, Sky values and Sky standard deviation.

```bash
$ ds9 -mecube r_detected.fits -zscale -zoom to fit
```

Flipping through the extensions in a FITS viewer, you will see that the first image (Sky-subtracted image) looks reasonable: there are no major artifacts due to bad Sky subtraction compared to the input. The second extension also seems reasonable with a large detection map that covers the whole of NGC5195, but also extends towards the bottom of the image where we actually see faint and diffuse signal in the input image.

Now try flipping between the DETECTIONS and SKY extensions. In the SKY extension, you’ll notice that there is still significant signal beyond the detected pixels. You can tell that this signal belongs to the galaxy because the far-right side of the image (away from M51) is dark (has lower values) and the brighter parts in the Sky image (with larger values) are just under the detections and follow a similar pattern.

The fact that signal from the galaxy remains in the SKY HDU shows that NoiseChisel can be optimized for a much better result. The SKY extension must not contain any light around the galaxy. Generally, any time your target is much larger than the tile size and the signal is very diffuse and extended at low signal-to-noise values (like this case), this will happen. Therefore, when there are large objects in the dataset, the best place to check the accuracy of your detection is the estimated Sky image.
When dominated by the background, noise has a symmetric distribution. However, signal is not symmetric (we don’t have negative signal). Therefore when non-constant\(^{27}\) signal is present in a noisy dataset, the distribution will be positively skewed. For a demonstration, see Figure 1 of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664). This skewness is a good measure of how much faint signal we have in the distribution. The skewness can be accurately measured by the difference in the mean and median (assuming no strong outliers): the more distant they are, the more skewed the dataset is. This important concept will be discussed more extensively in the next section (Section 2.3.3 [Skewness caused by signal and its measurement], page 83).

However, skewness is only a proxy for signal when the signal has structure (varies per pixel). Therefore, when it is approximately constant over a whole tile, or sub-set of the image, the constant signal’s effect is just to shift the symmetric center of the noise distribution to the positive and there won’t be any skewness (major difference between the mean and median). This positive\(^{28}\) shift that preserves the symmetric distribution is the Sky value. When there is a gradient over the dataset, different tiles will have different constant shifts/Sky-values, for example see Figure 11 of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664).

To make this very large diffuse/flat signal detectable, you will therefore need a larger tile to contain a larger change in the values within it (and improve number statistics, for less scatter when measuring the mean and median). So let’s play with the tessellation a little to see how it affects the result. In Gnuastro, you can see the option values (\texttt{--tilesize} in this case) by adding the \texttt{-P} option to your last command. Try running NoiseChisel with \texttt{-P} to see its default tile size.

You can clearly see that the default tile size is indeed much smaller than this (huge) galaxy and its tidal features. As a result, NoiseChisel was unable to identify the skewness within the tiles under the outer parts of M51 and NGC 5159 and the threshold has been over-estimated on those tiles. To see which tiles were used for estimating the quantile threshold (no skewness was measured), you can use NoiseChisel’s \texttt{--checkqthresh} option:

\begin{verbatim}
$ astnoisechisel r.fits -h0 --checkqthresh
\end{verbatim}

Did you see how NoiseChisel aborted after finding and applying the quantile thresholds? When you call any of NoiseChisel’s \texttt{--check*} options, by default, it will abort as soon as all the check steps have been written in the check file (a multi-extension FITS file). This allows you to focus on the problem you wanted to check as soon as possible (you can disable this feature with the \texttt{--continueaftercheck} option).

To optimize the threshold-related settings for this image, let’s play with this quantile threshold check image a little. Don’t forget that “\textit{Good statistical analysis is not a purely routine matter, and generally calls for more than one pass through the computer}” (Anscombe 1973, see Section 1.2 [Gnuastro manifesto: Science and its tools], page 2). A good scientist must have a good understanding of her tools to make a meaningful analysis. So don’t hesitate in playing with the default configuration and reviewing the manual when you have a new dataset (from a new instrument) in front of you. Robust data analysis is an art, therefore a good scientist must first be a good artist. So let’s open the check image as a multi-extension cube:
Chapter 2: Tutorials

$ ds9 -mecube r_qthresh.fits -zscale -cmap sls -zoom to fit

The first extension (called CONVOLVED) of r_qthresh.fits is the convolved input image where the threshold(s) is(are) defined (and later applied to). For more on the effect of convolution and thresholding, see Sections 3.1.1 and 3.1.2 of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664). The second extension (QTHRESH_ERODE) has a blank/white value for all the pixels of any tile that was identified as having significant signal. The other tiles have the measured threshold over them. The next two extensions (QTHRESH_NOERODE and QTHRESH_EXPAND) are the other two quantile thresholds that are necessary in NoiseChisel’s later steps. Every step in this file is repeated on the three thresholds.

Play a little with the color bar of the QTHRESH_ERODE extension, you clearly see how the non-blank tiles around NGC 5195 have a gradient. As one line of attack against discarding too much signal below the threshold, NoiseChisel rejects outlier tiles. Go forward by three extensions to VALUE1_NO_OUTLIER and you will see that many of the tiles over the galaxy have been removed in this step. For more on the outlier rejection algorithm, see the latter half of Section 7.1.4.3 [Quantifying signal in a tile], page 347.

Even though much of the galaxy’s footprint has been rejected as outliers, there are still tiles with signal remaining: play with the DS9 color-bar and you still see a gradient near the outer tidal feature of the galaxy. Before trying to correct this, let’s look at the other extensions of this check image. We will use a * as a wild-card that can be 1, 2 or 3. In the THRESH*_INTERP extensions, you see that all the blank tiles have been interpolated using their nearest neighbors (the relevant option here is --interpnumngb). In the following THRESH*_SMOOTH extensions, you can see the tile values after smoothing (configured with --smoothwidth option). Finally, in QTHRESH-APPLIED, you see the thresholded image: pixels with a value of 1 will be eroded later, but pixels with a value of 2 will pass the erosion step un-touched.

Let’s get back to the problem of optimizing the result. You have two strategies for detecting the outskirts of the merging galaxies: 1) Increase the tile size to get more accurate measurements of skewness. 2) Strengthen the outlier rejection parameters to discard more of the tiles with signal. Fortunately in this image we have a sufficiently large region on the right of the image that the galaxy doesn’t extend to. So we can use the more robust first solution. In situations where this doesn’t happen (for example if the field of view in this image was shifted to the left to have more of M51 and less sky) you are limited to a combination of the two solutions or just to the second solution.

Skipping convolution for faster tests: The slowest step of NoiseChisel is the convolution of the input dataset. Therefore when your dataset is large (unlike the one in this test), and you are not changing the input dataset or kernel in multiple runs (as in the tests of this tutorial), it is faster to do the convolution separately once (using Section 6.3 [Convolve], page 306) and use NoiseChisel’s --convolved option to directly feed the convolved image and avoid convolution. For more on --convolved, see Section 7.2.2.1 [NoiseChisel input], page 367.

To better identify the skewness caused by the flat NGC 5195 and M51 tidal features on the tiles under it, we have to choose a larger tile size. Let’s try a tile size of 100 by 100 pixels and inspect the check image.
$ \textit{astnoisechisel \texttt{r.fits -h0 --tilesize=100,100 --checkqthresh}}$

$ \textit{ds9 -mecube \texttt{r\_qthresh.fits -zscale -cmap sls -zoom to fit}}$

You can clearly see the effect of this increased tile size: the tiles are much larger and when you look into \texttt{VALUE1\_NO\_OUTLIER}, you see that all the tiles are nicely grouped on the right side of the image (the farthest from M51, where we don’t see a gradient in \texttt{QTHRESH\_ERODE}). Things look good now, so let’s remove \texttt{--checkqthresh} and let NoiseChisel proceed with its detection.

$ \textit{astnoisechisel \texttt{r.fits -h0 --tilesize=100,100}}$

$ \textit{ds9 -mecube \texttt{r\_detected.fits -zscale -cmap sls -zoom to fit}}$

The detected pixels of the \texttt{DETECTIONS} extension have expanded a little, but not as much. Also, the gradient in the \texttt{SKY} image is almost fully removed (and doesn’t fall over M51 anymore). However, on the bottom-right of the m51 detection, we see many holes gradually increasing in size. This hints that there is still signal out there. Let’s check the next series of detection steps by adding the \texttt{--checkdetection} option this time:

$ \textit{astnoisechisel \texttt{r.fits -h0 --tilesize=100,100 --checkdetection}}$

$ \textit{ds9 -mecube \texttt{r\_detcheck.fits -zscale -cmap sls -zoom to fit}}$

The output now has 16 extensions, showing every step that is taken by NoiseChisel. The first and second (\texttt{INPUT} and \texttt{CONVOLVED}) are clear from their names. The third (\texttt{THRESHOLDED}) is the thresholded image after finding the quantile threshold (last extension of the output of \texttt{--checkqthresh}). The fourth HDU (\texttt{ERODED}) is new: its the name-stake of NoiseChisel, or eroding pixels that are above the threshold. By erosion, we mean that all pixels with a value of 1 (above the threshold) that are touching a pixel with a value of 0 (below the threshold) will be flipped to zero (or “carved” out)\textsuperscript{29}. You can see its effect directly by going back and forth between the \texttt{THRESHOLDED} and \texttt{ERODED} extensions.

In the fifth extension (\texttt{OPENED-AND-LABELED}) the image is “opened”, which is a name for eroding once, then dilating (dilation is the inverse of erosion). This is good to remove thin connections that are only due to noise. Each separate connected group of pixels is also given its unique label here. Do you see how just beyond the large M51 detection, there are many smaller detections that get smaller as you go more distant? This hints at the solution: the default number of erosions is too much. Let’s see how many erosions take place by default (by adding \texttt{-P | grep erode} to the previous command)

$ \textit{astnoisechisel \texttt{r.fits -h0 --tilesize=100,100 -P | grep erode}}$

We see that the value of \texttt{erode} is 2. The default NoiseChisel parameters are primarily targeted to processed images (where there is correlated noise due to all the processing that has gone into the warping and stacking of raw images, see Section 2.2.11 \texttt{[NoiseChisel optimization for detection]}, page 46). In those scenarios 2 erosions are commonly necessary. But here, we have a single-exposure image where there is no correlated noise (the pixels aren’t mixed). So let’s see how things change with only one erosion:

$ \textit{astnoisechisel \texttt{r.fits -h0 --tilesize=100,100 --erode=1 \ --checkdetection}}$

$ \textit{ds9 -mecube \texttt{r\_detcheck.fits -zscale -cmap sls -zoom to fit}}$

\textsuperscript{29} Pixels with a value of 2 are very high signal-to-noise pixels, they are not eroded, to preserve sharp and bright sources.
Looking at the **OPENED-AND-LABELLED** extension again, we see that the main/large detection is now much larger than before. While the immediately-outer connected regions are still present, they have decreased dramatically, so we can pass this step.

After the **OPENED-AND-LABELLED** extension, NoiseChisel goes onto finding false detections using the undetected pixels. The process is fully described in Section 3.1.5. (Defining and Removing False Detections) of arXiv:1505.01664 (https://arxiv.org/pdf/1505.01664.pdf). Please compare the extensions to what you read there and things will be very clear. In the last HDU (**DETECTION-FINAL**), we have the final detected pixels that will be used to estimate the Sky and its Standard deviation. We see that the main detection has indeed been detected very far out, so let’s see how the full NoiseChisel will estimate the Sky and its standard deviation (by removing --checkdetection):

```
$ astnoisechisel r.fits -h0 --tilesize=100,100 --erode=1
$ ds9 -mecube r_detected.fits -zscale -cmap sls -zoom to fit
```

The **DETECTIONS** extension of r_detected.fits closely follows what the **DETECTION-FINAL** of the check image (looks good!). If you go ahead to the **SKY** extension, things still look good. But it can still be improved.

Look at the **DETECTIONS** again, you will see the right-ward edges of M51’s detected pixels have many “holes” that are fully surrounded by signal (value of 1) and the signal stretches out in the noise very thinly (the size of the holes increases as we go out). This suggests that there is still undetected signal and that we can still dig deeper into the noise.

With the --**detgrowquant** option, NoiseChisel will “grow” the detections in to the noise. Its value is the ultimate limit of the growth in units of quantile (between 0 and 1). Therefore --**detgrowquant=1** means no growth and --**detgrowquant=0.5** means an ultimate limit of the Sky level (which is usually too much and will cover the whole image!). See Figure 2 of arXiv:1909.11230 (https://arxiv.org/pdf/1909.11230.pdf) for more on this option. Try running the previous command with various values (from 0.6 to higher values) to see this option’s effect on this dataset. For this particularly huge galaxy (with signal that extends very gradually into the noise), we’ll set it to 0.75:

```
$ astnoisechisel r.fits -h0 --tilesize=100,100 --erode=1
  --detgrowquant=0.75
$ ds9 -mecube r_detected.fits -zscale -cmap sls -zoom to fit
```

Beyond this level (smaller --**detgrowquant** values), you see many of the smaller background galaxies (towards the right side of the image) starting to create thin spider-leg-like features, showing that we are following correlated noise for too much. Please try it for your self by changing it to 0.6 for example.

When you look at the **DETECTIONS** extension of the command shown above, you see the wings of the galaxy being detected much farther out. But you also see many holes which are clearly just caused by noise. After growing the objects, NoiseChisel also allows you to fill such holes when they are smaller than a certain size through the --**detgrowmaxholesize** option. In this case, a maximum area/size of 10,000 pixels seems to be good:

```
$ astnoisechisel r.fits -h0 --tilesize=100,100 --erode=1
  --detgrowquant=0.75 --detgrowmaxholesize=10000
$ ds9 -mecube r_detected.fits -zscale -cmap sls -zoom to fit
```

When looking at the raw input image (which is very “shallow”: less than a minute exposure!), you don’t see anything so far out of the galaxy. You might just think to yourself
that “this is all noise, I have just dug too deep and I’m following systematics”! If you feel like this, have a look at the deep images of this system in Watkins et al. [2015] (https://arxiv.org/abs/1501.04599), or a 12 hour deep image of this system (with a 12-inch telescope): https://i.redd.it/jfqgpqg0hfk11.jpg.

In these deeper images you clearly see how the outer edges of the M51 group follow this exact structure, below in Section 2.3.5 [Achieved surface brightness level], page 92, we’ll measure the exact level.

As the gradient in the SKY extension shows, and the deep images cited above confirm, the galaxy’s signal extends even beyond this. But this is already far deeper than what most (if not all) other tools can detect. Therefore, we’ll stop configuring NoiseChisel at this point in the tutorial and let you play with the other options a little more, while reading more about it in the papers (Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664) and Akhlaghi [2019] (https://arxiv.org/abs/1909.11230)) and Section 7.2 [NoiseChisel], page 362. When you do find a better configuration feel free to contact us for feedback. Don’t forget that good data analysis is an art, so like a sculptor, master your chisel for a good result.

To avoid typing all these options every time you run NoiseChisel on this image, you can use Gnuastro’s configuration files, see Section 4.2 [Configuration files], page 169. For an applied example of setting/using them, see Section 2.2.8 [Option management and configuration files], page 40.

This NoiseChisel configuration is NOT GENERIC: Don’t use the configuration derived above, on another instrument’s image blindly. If you are unsure, just use the default values. As you saw above, the reason we chose this particular configuration for NoiseChisel to detect the wings of the M51 group was strongly influenced by the noise properties of this particular image. Remember Section 2.2.11 [NoiseChisel optimization for detection], page 46, where we looked into the very deep XDF image which had strong correlated noise?

As long as your other images have similar noise properties (from the same data-reduction step of the same instrument), you can use your configuration on any of them. But for images from other instruments, please follow a similar logic to what was presented in these tutorials to find the optimal configuration.

Smart NoiseChisel: As you saw during this section, there is a clear logic behind the optimal parameter value for each dataset. Therefore, we plan to capabilities to (optionally) automate some of the choices made here based on the actual dataset, please join us in doing this if you are interested. However, given the many problems in existing “smart” solutions, such automatic changing of the configuration may cause more problems than they solve. So even when they are implemented, we would strongly recommend quality checks for a robust analysis.

2.3.3 Skewness caused by signal and its measurement

In the previous section (Section 2.3.2 [NoiseChisel optimization], page 78) we showed how to customize NoiseChisel for a single-exposure SDSS image of the M51 group. During
the customization, we also discussed the skewness caused by signal. In the next section (Section 2.3.4 [Image surface brightness limit], page 88), we will use this to measure the surface brightness limit of the image. However, to better understand NoiseChisel and also, the image surface brightness limit, understanding the skewness caused by signal, and how to measure it properly are very important. Therefore now that we have separated signal from noise, let’s pause for a moment and look into skewness, how signal creates it, and find the best way to measure it.

Let’s start masking all the detected pixels found at the end of the previous section (Section 2.3.2 [NoiseChisel optimization], page 78) and having a look at the noise distribution with Gnuastro’s Arithmetic and Statistics programs as shown below (while visually inspecting the masked image with DS9 in the middle).

$$\text{astarithmetic r_detected.fits -hINPUT-NO-SKY set-in} \ \ r\text{_detected.fits -hDETECTIONS set-det} \ \ \text{in det nan where -odet-masked.fits}$$

$\text{ds9 det-masked.fits}$

$\text{aststatistics det-masked.fits}$

You will see that Gnuastro’s Statistics program prints an ASCII histogram when no option is given (it is shown below). This is done to give you a fast and easy view of the distribution of values in the dataset (pixels in an image, or rows in a table’s column).

-------
Input: det-masked.fits (hdu: 1)
-------

Number of elements: 903920
Minimum: -0.113543
Maximum: 0.130339
Median: -0.00216306
Mean: -0.0001893073877

Standard deviation: 0.02569057188

-------

Histogram:

This histogram shows a roughly symmetric noise distribution, so let’s have a look at its skewness. The most commonly used definition of skewness is known as the “Pearson’s first skewness coefficient”. It measures the difference between the mean and median, in units of the standard deviation (STD):
Skewness \equiv \frac{(\text{mean} - \text{median})}{\text{STD}}

The logic behind this definition is simple: as more signal is added to the same pixels that originally only have raw noise (skewness is increased), the mean shifts to the positive faster than the median, so the distance between the mean and median should increase. Let’s measure the skewness (as defined above) over the image without any signal. It’s very easy with Gnuastro’s Statistics program (and piping the output to AWK):

```
$ aststatistics det-masked.fits --mean --median --std \
   | awk '{print ($1-$2)/$3}'
0.0768279
```

We see that the mean and median are only 0.08\sigma \ (\text{rounded}) away from each other (which is very close)! All pixels with significant signal are masked, so this is expected, and everything is fine. Now, let’s check the pixel distribution of the sky-subtracted input (where pixels with significant signal remain, and are not masked):

```
$ ds9 r_detected.fits
$ aststatistics r_detected.fits -hINPUT-NO-SKY
-------
Input: r_detected.fits (hdu: INPUT-NO-SKY)
Unit: nanomaggy
-------
Number of elements: 3049472
Minimum: -0.113543
Maximum: 159.25
Median: 0.0241158
Mean: 0.1057885317
Standard deviation: 0.698167489
-------
Histogram:
|*
|*
|*
|*
|*
|*
|*
|*
|*
|*
|*
|*
|*

Comparing the distributions above, you can see that the minimum value of the image has not changed because we have not masked the minimum values. However, as expected, the maximum value of the image has changed (from 0.13 to 159.25). This is clearly evident
from the ASCII histogram: the distribution is very elongated because the galaxy inside the image is extremely bright.

Now, let’s limit the displayed information with the `--lessthan=0.13` option of Statistics as shown below (to only use values less than 0.13; the maximum of the image where all signal is masked).

```
$ aststatistics r_detected.fits -hINPUT-NO-SKY --lessthan=0.13
```

```
Input: r_detected.fits (hdu: INPUT-NO-SKY)
Range: up to (exclusive) 0.13.
Unit: nanomaggy

Number of elements: 2531949
Minimum: -0.113543
Maximum: 0.126233
Median: 0.0137138
Mean: 0.01735551527
Standard deviation: 0.03590550597
```

```
Histogram:
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
|
```

The improvement is obvious: the ASCII histogram better shows the pixel values near the noise level. We can now compare with the distribution of `det-masked.fits` that we found earlier. The ASCII histogram of `det-masked.fits` was approximately symmetric, while this is asymmetric in this range, especially in outer (to the right, or positive) direction. The heavier right-side tail is a clear visual demonstration of skewness that is cased by the signal in the un-masked image.

Having visually confirmed the skewness, let’s quantify it with Pearson’s first skewness coefficient. Like before, we can simply use Gnuastro’s Statistics and AWK for the measurement and calculation:

```
$ aststatistics r_detected.fits --mean --median --std \  
   | awk '{print ($1-$2)/$3}'
```

```
0.116982
```

The difference between the mean and median is now approximately 0.12σ. This is larger than the skewness of the masked image (which was approximately 0.08σ). At a glance
(only looking at the numbers), it seems that there is not much difference between the two
distributions. However, visually looking at the non-masked image, or the ASCII histogram,
you would expect the quantified skewness to be much larger than that of the masked image,
but that hasn’t happened! Why is that?

The reason is that the presence of signal doesn’t only shift the mean and median, it also
increases the standard deviation! To see this for yourself, compare the standard dev-
iation of `det-masked.fits` (which was approximately 0.025) `r_detected.fits` (without
`--lessthan`; which was approximately 0.699). The latter is almost 28 times larger!

This happens because the standard deviation is defined only in a symmetric (and Gaus-
sian) distribution. In a non-Gaussian distribution, the standard deviation is poorly defined
and isn’t a good measure of “width”. Since Pearson’s first skewness coefficient is defined
in units of the standard deviation, this very large increase in the standard deviation has
hidden the much increased distance between the mean and median after adding signal.

We therefore need a better unit or scale to quantify the distance between the mean and
median. A unit that is less affected by skewness or outliers. One solution that we have
found to be very useful is the quantile units or quantile scale. The quantile scale is defined
by first sorting the dataset (which has \( N \) elements). If we want the quantile of a value \( V \)
in a distribution, we first find the nearest data element to \( V \) in the sorted dataset. Let’s
assume the nearest element is the \( i \)-th element, counting from 0, after sorting. The quantile
of \( V \) in that distribution is then defined as \( i/(N - 1) \) (which will have a value between 0
and 1).

The quantile of the median is obvious from its definition: 0.5. This is because the median
is defined to be the middle element of the distribution after sorting. We can therefore define
skewness as the quantile of the mean (\( q_m \)). If \( q_m \sim 0.5 \) (the median), then the distribution
(of signal blended in noise) is symmetric (possibly Gaussian, but the functional form is
irrelevant here). A larger value for \( |q_m - 0.5| \) quantifies a more skewed the distribution.
Furthermore, a \( q_m > 0.5 \) signifies a positive skewness, while \( q_m < 0.5 \) signifies a negative
skewness.

Let’s put this definition to a test on the same two images we have already created.
Fortunately Gnuastro’s Statistics program has the `--quantofmean` option to easily calculate
\( q_m \) for you. So testing is easy:

\[
\begin{align*}
\$ & \text{aststatistics det-masked.fits --quantofmean} \\
& 0.51295636 \\
\$ & \text{aststatistics r_detected.fits -hINPUT-NO-SKY --quantofmean} \\
& 0.8105163158
\end{align*}
\]

The two quantiles of mean are now very distinctly different (0.51 and 0.81): differing by
about 0.3 (on a scale of 0 to 1)! Recall that when defining skewness with Pearson’s first skew-
ness coefficient, their difference was negligible (0.04\( \sigma \))! You can now better appreciate why
we discussed quantile so extensively in Section 2.3.2 [NoiseChisel optimization], page 78. In
case you would like to know more about the usage of the quantile of the mean in Gnuastro,
please see Section 7.1.4.3 [Quantifying signal in a tile], page 347, or watch this video demon-
stration: https://peertube.stream/w/35b7c398-9fd7-4bfc-8911-1e01c5124585.
2.3.4 Image surface brightness limit

When your science is related to extended emission (like the example here) and you are presenting your results in a scientific conference, usually the first thing that someone will ask (if you don’t explicitly say it!), is the dataset’s surface brightness limit (a standard measure of the noise level), and your target’s surface brightness (a measure of the signal, either in the center or outskirts, depending on context). For more on the basics of these important concepts please see Section 7.4.3 [Quantifying measurement limits], page 398. So in this section of the tutorial, we’ll measure these values for this image and this target.

Before measuring the surface brightness limit, let’s see how reliable our detection was. In other words, let’s see how “clean” our noise is (after masking all detections, as described previously in Section 2.3.3 [Skewness caused by signal and its measurement], page 83)

$ aststatistics det-masked.fits --quantofmean
0.5111848629

Showing that the mean is indeed very close to the median, although just about 1 quantile larger. As we saw in Section 2.3.2 [NoiseChisel optimization], page 78, a very small residual signal still remains in the undetected regions and this very small difference is a quantitative measure of that undetected signal. It was up to you as an exercise to improve it, so we’ll continue with this dataset.

The surface brightness limit of the image can be measured from the masked image and the equation in Section 7.4.3 [Quantifying measurement limits], page 398. Let’s do it for a 3σ surface brightness limit over an area of 25arcsec²:

$ nsigma=3
$ zeropoint=22.5
$ areaarcsec2=25
$ std=$(aststatistics det-masked.fits --sigclip-std)
$ pixarcsec2=$(astfits det-masked.fits --pixelscale --quiet | awk '{print $3*3600*3600}')
$ astarithmetic --quiet $nsigma $std x $areaarcsec2 $pixarcsec2 x sqrt / $zeropoint counts-to-mag
26.0241

The customizable steps above are good for any type of mask. For example your field of view may contain a very deep part so you need to mask all the shallow parts as well as the detections before these steps. But when your image is flat (like this), there is a much simpler method to obtain the same value through MakeCatalog (when the standard deviation image is made by NoiseChisel). NoiseChisel has already calculated the minimum (MINSTD), maximum (MAXSTD) and median (MEDSTD) standard deviation within the tiles during its processing and has stored them as FITS keywords within the SKY_STD HDU. You can see them by piping all the keywords in this HDU into grep. In Grep, each ‘.’ represents one character that can be anything so M..STD will match all three keywords mentioned above.

$ astfits r_detected.fits --hdu=SKY_STD | grep 'M..STD'

The MEDSTD value is very similar to the standard deviation derived above, so we can safely use it instead of having to mask and run Statistics. In fact, MakeCatalog also uses
this keyword and will report the dataset’s $n\sigma$ surface brightness limit as keywords in the output (not as measurement columns, since it is related to the noise, not labeled signal):

```
$ astmkcatalog r_detected.fits -hDETECTIONS --output=sbl.fits \\
   --forcereadstd --ids
```

Before looking into the measured surface brightness limits, let’s review some important points about this call to MakeCatalog first:

- We are only concerned with the noise (not the signal), so we don’t ask for any further measurements, because they can un-necessarily slow it down. However, MakeCatalog requires at least one column, so we’ll only ask for the --ids column (which doesn’t need any measurement!). The output catalog will therefore have a single row and a single column, with 1 as its value\(^{31}\).

- If we don’t ask for any noise-related column (for example the signal-to-noise ratio column with --sn, among other noise-related columns), MakeCatalog is not going to read the noise standard deviation image (again, to speed up its operation when it is redundant). We are thus using the --forcereadstd option (short for “force read standard deviation image”) here so it is ready for the surface brightness limit measurements that are written as keywords.

With the command below you can see all the keywords that were measured with the table. Notice the group of keywords that are under the “Surface brightness limit (SBL)” title.

```
$ astfits sbl.fits -h1
```

Since all the keywords of interest here start with SBL, we can get a more cleaner view with this command.

```
$ astfits sbl.fits -h1 | grep ^SBL
```

Notice how the SBLSTD has the same value as NoiseChisel’s MEDSTD above. Using SBLSTD, MakeCatalog has determined the $n\sigma$ surface brightness limiting magnitude in these header keywords. The multiple of $\sigma$, or $n$, is the value of the SBLNSIG keyword which you can change with the --sfmagnsigma option. The surface brightness limiting magnitude within a pixel (SBLNSIG) and within a pixel-agnostic area of SBLAREA arcsec\(^2\) are stored in SBLMAG.

You will notice that the two surface brightness limiting magnitudes above have values around 3 and 4 (which is not correct!). This is because we haven’t given a zero point magnitude to MakeCatalog, so it uses the default value of 0. SDSS image pixel values are calibrated in units of “nanomaggy” which are defined to have a zero point magnitude of 22.5\(^{32}\). So with the first command below we give the zero point value and with the second we can see the surface brightness limiting magnitudes with the correct values (around 25 and 26)

```
$ astmkcatalog r_detected.fits -hDETECTIONS --zeropoint=22.5 \\
   --output=sbl.fits --forcereadstd --ids
```

```
$ astfits sbl.fits -h1 | grep ^SBL
```

---

\(^{31}\) Recall that NoiseChisel’s output is a binary image: 0-valued pixels are noise and 1-valued pixel are signal. NoiseChisel doesn’t identify sub-structure over the signal, this is the job of Segment, see Section 2.3.6 [Extract clumps and objects (Segmentation)], page 94.

\(^{32}\) From [https://www.sdss.org/dr12/algorithms/magnitudes](https://www.sdss.org/dr12/algorithms/magnitudes)
As you see from SBLNSIG and SBLAREA, the default multiple of sigma is 1 and the default area is 1 arcsec$^2$. Usually higher values are used for these two parameters. Following the manual example we did above, you can ask for the multiple of sigma to be 3 and the area to be 25 arcsec$^2$:

```bash
$ astmkcatalog r_detected.fits -hDETECTIONS --zeropoint=22.5 \  --output=sbl.fits --sfmagarea=25 --sfmagnsigma=3 \  --forcereadstd --ids

$ astfits sbl.fits -h1 | awk '/^SBLMAG /{print $3}'

```

26.02296

You see that the value is identical to the custom surface brightness limiting magnitude we measured above (a difference of 0.00114 magnitudes is negligible and hundreds of times larger than the typical errors in the zero point magnitude or magnitude measurements). But it is much more easier to have MakeCatalog do this measurement, because these values will be appended (as keywords) into your final catalog of objects within that image.

**Custom STD for MakeCatalog’s Surface brightness limit:** You can manually change/set the value of the MEDSTD keyword in your input STD image with Section 5.1 [Fits], page 194:

```bash
$ std=$(aststatistics masked.fits --sigclip-std)

$ astfits noisechisel.fits -hSKY_STD --update=MEDSTD,$std

```

With this change, MakeCatalog will use your custom standard deviation for the surface brightness limit. This is necessary in scenarios where your image has multiple depths and during your masking, you also mask the shallow regions (as well as the detections of course).

We have successfully measured the image’s 3σ surface brightness limiting magnitude over 25 arcsec$^2$. However, as discussed in Section 7.4.3 [Quantifying measurement limits], page 398, this value is just an extrapolation of the per-pixel standard deviation. Issues like correlated noise will cause the real noise over a large area to be different. So for a more robust measurement, let’s use the upper-limit magnitude of similarly sized region. For more on the upper-limit magnitude, see the respective item in Section 7.4.3 [Quantifying measurement limits], page 398.

In summary, the upper-limit measurements involve randomly placing the footprint of an object in undetected parts of the image many times. This results in a random distribution of brightness measurements, the standard deviation of that distribution is then converted into magnitudes. To be comparable with the results above, let’s make a circular aperture that has an area of 25 arcsec$^2$ (thus with a radius of 2.82095 arcsec).

```
zeropoint=22.5
r_arcsec=2.82095

## Convert the radius (in arcseconds) to pixels.
## Make circular aperture at pixel (100,100) position is irrelevant.
```

```bash
r_pixel=$(astfits r_detected.fits --pixelscale -q \  | awk '{print '$r_arcsec'/($1*3600)}')

## Convert the radius (in arcseconds) to pixels.
```

```bash
## Make circular aperture at pixel (100,100) position is irrelevant.
echo "1 100 100 5 $r_pixel 0 0 1 1 1" \  | astmkprof --background=r_detected.fits \
```
--clearcanvas --mforflatpix --type=uint8 \
--output=lab.fits

## Do the upper-limit measurement, ignoring all NoiseChisel's 
## detections as a mask for the upper-limit measurements.
astmkcatalog lab.fits -h1 --zeropoint=$zeropoint -osbl.fits \
   --sfmagarea=25 --sfmagnsigma=3 --forcereadstd \
   --valuesfile=r_detected.fits --valueshdu=INPUT-NO-SKY \
   --upmaskfile=r_detected.fits --upmaskhdu=DETECTIONS \
   --upnsigma=3 --checkuplim=1 --upnum=1000 \
   --ids --upperlimitsb

The sbl.fits catalog now contains the upper-limit surface brightness for a circle with 
an area of 25 arcsec$^2$. You can check the value with the command below, but the great 
thing is that now you have both the surface brightness limiting magnitude in the headers 
discussed above, and the upper-limit surface brightness within the table. You can also 
add more profiles with different shapes and sizes if necessary. Of course, you can also 
use --upperlimitsb in your actual science objects and clumps to get an object-specific or 
clump-specific value.

$ asttable sbl.fits -cUPPERLIMIT_SB
25.9119

You will get a slightly different value from the command above. In fact, if you run the 
MakeCatalog command again and look at the measured upper-limit surface brightness, it 
will be slightly different with your first trial! Please try exactly the same MakeCatalog 
command above a few times to see how it changes.

This is because of the random factor in the upper-limit measurements: every time you 
run it, different random points will be checked, resulting in a slightly different distribution. 
You can decrease the random scatter by increasing the number of random checks (for 
example setting --upnum=100000, compared to 1000 in the command above). But this 
will be slower and the results won’t be exactly reproducible. The only way to ensure you 
get an identical result later is to fix the random number generator function and seed like the 
command below. This is a very important point regarding any statistical process involving 
random numbers, please see Section 8.2.1.4 [Generating random numbers], page 464.

    export GSL_RNG_TYPE=ranlxs1
    export GSL_RNG_SEED=1616493518
    astmkcatalog lab.fits -h1 --zeropoint=$zeropoint -osbl.fits \
       --sfmagarea=25 --sfmagnsigma=3 --forcereadstd \
       --valuesfile=r_detected.fits --valueshdu=INPUT-NO-SKY \
       --upmaskfile=r_detected.fits --upmaskhdu=DETECTIONS \
       --upnsigma=3 --checkuplim=1 --upnum=1000 \
       --ids --upperlimitsb --envseed

But where do all the random apertures of the upper-limit measurement fall on the image? 
It is good to actually inspect their location to get a better understanding for the process and 
also detect possible bugs/biases. When MakeCatalog is run with the --checkuplim option,

---

You can use any integer for the seed. One recommendation is to run MakeCatalog without --envseed 
once and use the randomly generated seed that is printed on the terminal.
it will print all the random locations and their measured brightness as a table in a file with
the suffix _upcheck.fits. With the first command below you can use Gnuastro’s asttable
and astscript-ds9-region to convert the successful aperture locations into a DS9 region
file, and with the second can load the region file into the detections and sky-subtracted
image to visually see where they are.

```bash
## Create a DS9 region file from the check table (activated
## with '--checkuplim')
asttable lab_upcheck.fits --noblank=RANDOM_SUM \ 
    | astscript-ds9-region -c1,2 --mode=img \ 
    --radius=$r_pixel
```

## Have a look at the regions in relation with NoiseChisel's
detections.
ds9 r_detected.fits[INPUT-NO-SKY] -regions load ds9.reg
ds9 r_detected.fits[DETECTIONS] -regions load ds9.reg

In this example, we were looking at a single-exposure image that has no correlated
noise. Because of this, the surface brightness limit and the upper-limit surface brightness
are very close. They will have a bigger difference on deep datasets with stronger correlated
noise (that are the result of stacking many individual exposures). As an exercise, please
try measuring the upper-limit surface brightness level and surface brightness limit for the
deep HST data that we used in the previous tutorial (Section 2.2 [General program usage
tutorial], page 29).

### 2.3.5 Achieved surface brightness level

In Section 2.3.2 [NoiseChisel optimization], page 78, we customized NoiseChisel for a single-
exposure SDSS image of the M51 group and in Section 2.3.4 [Image surface brightness limit],
page 88, we measured the surface brightness limit and the upper-limit surface brightness
level (which are both measures of the noise level). In this section, let’s do some mea-
surements on the outer-most edges of the M51 group to see how they relate to the noise
measurements found in the previous section.

For this measurement, we’ll need to estimate the average flux on the outer edges of the
detection. Fortunately all this can be done with a few simple commands using Section 6.2
[Arithmetic], page 272, and Section 7.4 [MakeCatalog], page 392. First, let’s separate each
detected region, or give a unique label/counter to all the connected pixels of NoiseChisel’s
detection map with the command below. Recall that with the set- operator, the popped
operand will be given a name (det in this case) for easy usage later.

```bash
$ astarithmetic r_detected.fits -hDETECTIONS set-det \ 
    det 2 connected-components -olabeled.fits
```

You can find the label of the main galaxy visually (by opening the image and hovering
your mouse over the M51 group’s label). But to have a little more fun, let’s do this auto-
matically (which is necessary in a general scenario). The M51 group detection is by far
the largest detection in this image, this allows us to find its ID/label easily. We’ll first run
MakeCatalog to find the area of all the labels, then we’ll use Table to find the ID of the
largest object and keep it as a shell variable (id):

```bash
# Run MakeCatalog to find the area of each label.
```
$ astmkcatalog labeled.fits --ids --geoarea -h1 -ocat.fits

## Sort the table by the area column.
$ asttable cat.fits --sort=AREA_FULL

## The largest object, is the last one, so we'll use '--tail'.
$ asttable cat.fits --sort=AREA_FULL --tail=1

## We only want the ID, so let's only ask for that column:
$ asttable cat.fits --sort=AREA_FULL --tail=1 --column=OBJ_ID

## Now, let's put this result in a variable (instead of printing)
$id=$(asttable cat.fits --sort=AREA_FULL --tail=1 --column=OBJ_ID)

## Just to confirm everything is fine.
$ echo $id

We can now use the $id variable to reject all other detections:

$ astarithmetic labeled.fits $id eq -oonly-m51.fits

Open the image and have a look. To separate the outer edges of the detections, we'll need to “erode” the M51 group detection. So in the same Arithmetic command as above, we'll erode three times (to have more pixels and thus less scatter), using a maximum connectivity of 2 (8-connected neighbors). We'll then save the output in eroded.fits.

$ astarithmetic labeled.fits $id eq 2 erode 2 erode 2 erode \ -oeroded.fits

In labeled.fits, we can now set all the 1-valued pixels of eroded.fits to 0 using Arithmetic’s where operator added to the previous command. We'll need the pixels of the M51 group in labeled.fits two times: once to do the erosion, another time to find the outer pixel layer. To do this (and be efficient and more readable) we'll use the set-i operator (to give this image the name ‘i’). In this way we can use it any number of times afterwards, while only reading it from disk and finding M51’s pixels once.

$ astarithmetic labeled.fits $id eq set-i i \ i 2 erode 2 erode 2 erode 0 where -oedge.fits

Open the image and have a look. You'll see that the detected edge of the M51 group is now clearly visible. You can use edge.fits to mark (set to blank) this boundary on the input image and get a visual feeling of how far it extends:

$ astarithmetic r.fits -h0 edge.fits nan where -oedge-masked.fits

To quantify how deep we have detected the low-surface brightness regions (in units of signal to-noise ratio), we'll use the command below. In short it just divides all the non-zero pixels of edge.fits in the Sky subtracted input (first extension of NoiseChisel’s output) by the pixel standard deviation of the same pixel. This will give us a signal-to-noise ratio image. The mean value of this image shows the level of surface brightness that we have achieved. You can also break the command below into multiple calls to Arithmetic and create temporary files to understand it better. However, if you have a look at Section 6.2.1
[Reverse polish notation], page 272, and Section 6.2.3 [Arithmetic operators], page 276, you should be able to easily understand what your computer does when you run this command

```bash
$ astarithmetic edge.fits -h1 set-edge \
  r_detected.fits -hSKY_STD set-skystd \
  r_detected.fits -hINPUT-NO-SKY set-skysub \
  skysub skystd / edge not nan where meanvalue --quiet
```

We have thus detected the wings of the M51 group down to roughly 1/3rd of the noise level in this image which is a very good achievement! But the per-pixel S/N is a relative measurement. Let’s also measure the depth of our detection in absolute surface brightness units; or magnitudes per square arc-seconds (see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394). We’ll also ask for the S/N and magnitude of the full edge we have defined. Fortunately doing this is very easy with Gnuastro’s MakeCatalog:

```bash
$ astmkcatalog edge.fits -h1 --valuesfile=r_detected.fits \
  --zeropoint=22.5 --ids --surfacebrightness --sn \
  --magnitude
$ asttable edge_cat.fits
1 25.6971 55.2406 15.8994
```

We have thus reached an outer surface brightness of 25.70 magnitudes/arcsec² (second column in `edge_cat.fits`) on this single exposure SDSS image! This is very similar to the surface brightness limit measured in Section 2.3.4 [Image surface brightness limit], page 88, (which is a big achievement!). But another point in the result above is very interesting: the total S/N of the edge is 55.24 with a total edge magnitude of 15.90!!! This very large for such a faint signal (recall that the mean S/N per pixel was 0.32) and shows a very important point in the study of galaxies: While the per-pixel signal in their outer edges may be very faint (and invisible to the eye in noise), a lot of signal hides deeply buried in the noise.

In interpreting this value, you should just have in mind that NoiseChisel works based on the contiguity of signal in the pixels. Therefore the larger the object, the deeper NoiseChisel can carve it out of the noise (for the same outer surface brightness). In other words, this reported depth, is the depth we have reached for this object in this dataset, processed with this particular NoiseChisel configuration. If the M51 group in this image was larger/smaller than this (the field of view was smaller/larger), or if the image was from a different instrument, or if we had used a different configuration, we would go deeper/shallower.

### 2.3.6 Extract clumps and objects (Segmentation)

In Section 2.3.2 [NoiseChisel optimization], page 78, we found a good detection map over the image, so pixels harboring signal have been differentiated from those that don’t. For noise-related measurements like the surface brightness limit, this is fine. However, after finding

---

34 `edge.fits` (extension 1) is a binary (0 or 1 valued) image. Applying the `not` operator on it, just flips all its pixels (from 0 to 1 and vice-versa). Using the `where` operator, we are then setting all the newly 1-valued pixels (pixels that aren’t on the edge) to NaN/blank in the sky-subtracted input image (`r_detected.fits`, extension INPUT-NO-SKY, which we call skysub). We are then dividing all the non-blank pixels (only those on the edge) by the sky standard deviation (`r_detected.fits`, extension SKY_STD, which we called skystd). This gives the signal-to-noise ratio (S/N) for each of the pixels on the boundary. Finally, with the `meanvalue` operator, we are taking the mean value of all the non-blank pixels and reporting that as a single number.

35 You can run MakeCatalog on `only-m51.fits` instead of `edge.fits` to see the full magnitude of the M51 group in this image.
the pixels with signal, you are most likely interested in knowing the sub-structure within them. For example how many star forming regions (those bright dots along the spiral arms) of M51 are within this image? What are the colors of each of these star forming regions? In the outer most wings of M51, which pixels belong to background galaxies and foreground stars? And many more similar questions. To address these questions, you can use Section 7.3 [Segment], page 381, to identify all the “clumps” and “objects” over the detection.

$ astsegment r_detected.fits --output=r_segmented.fits
$ ds9 -mecube r_segmented.fits -cmap sls -zoom to fit -scale limits 0 2

Open the output r_segmented.fits as a multi-extension data cube with the second command above and flip through the first and second extensions, zoom-in to the spiral arms of M51 and see the detected clumps (all pixels with a value larger than 1 in the second extension). To optimize the parameters and make sure you have detected what you wanted, we recommend to visually inspect the detected clumps on the input image.

For visual inspection, you can make a simple shell script like below. It will first call MakeCatalog to estimate the positions of the clumps, then make an SAO DS9 region file and open ds9 with the image and region file. Recall that in a shell script, the numeric variables (like $1, $2, and $3 in the example below) represent the arguments given to the script. But when used in the AWK arguments, they refer to column numbers.

To create the shell script, using your favorite text editor, put the contents below into a file called check-clumps.sh. Recall that everything after a # is just comments to help you understand the command (so read them!). Also note that if you are copying from the PDF version of this book, fix the single quotes in the AWK command.

```bash
#!/bin/bash
set -e # Stop execution when there is an error.
set -u # Stop execution when a variable is not initialized.

# Run MakeCatalog to write the coordinates into a FITS table.
# Default output is '$1_cat.fits'.
astmkcatalog $1.fits --clumpscat --ids --ra --dec

# Use Gnuastro's Table and astscript-ds9-region to build the DS9
# region file (a circle of radius 1 arcseconds on each point).
asttable $1"_cat.fits" -hCLUMPS -cRA,DEC \ 
  | astscript-ds9-region -c1,2 --mode=wcs --radius=1 \ 
    --output=$1.reg

# Show the image (with the requested color scale) and the region file.
ds9 -geometry 1800x3000 -mecube $1.fits -zoom to fit \ 
  -scale limits $2 $3 -regions load all $1.reg

# Clean up (delete intermediate files).
rm $1"_cat.fits" $1.reg
```

Finally, you just have to activate the script’s executable flag with the command below. This will enable you to directly/easily call the script as a command.
$ chmod +x check-clumps.sh

This script doesn’t expect the .fits suffix of the input’s filename as the first argument. Because the script produces intermediate files (a catalog and DS9 region file, which are later deleted). However, we don’t want multiple instances of the script (on different files in the same directory) to collide (read/write to the same intermediate files). Therefore, we have used suffixes added to the input’s name to identify the intermediate files. Note how all the $1 instances in the commands (not within the AWK command$^{36}$) are followed by a suffix. If you want to keep the intermediate files, put a # at the start of the last line.

The few, but high-valued, bright pixels in the central parts of the galaxies can hinder easy visual inspection of the fainter parts of the image. With the second and third arguments to this script, you can set the numerical values of the color map (first is minimum/black, second is maximum/white). You can call this script with any$^{37}$ output of Segment (when --rawoutput is not used) with a command like this:

$ ./check-clumps.sh r_segmented -0.1 2

Go ahead and run this command. You will see the intermediate processing being done and finally it opens SAO DS9 for you with the regions superimposed on all the extensions of Segment’s output. The script will only finish (and give you control of the command-line) when you close DS9. If you need your access to the command-line before closing DS9, add a & after the end of the command above.

While DS9 is open, slide the dynamic range (values for black and white, or minimum/maximum values in different color schemes) and zoom into various regions of the M51 group to see if you are satisfied with the detected clumps. Don’t forget that through the “Cube” window that is opened along with DS9, you can flip through the extensions and see the actual clumps also. The questions you should be asking your self are these: 1) Which real clumps (as you visually feel) have been missed? In other words, is the completeness good? 2) Are there any clumps which you feel are false? In other words, is the purity good?

Note that completeness and purity are not independent of each other, they are anti-correlated: the higher your purity, the lower your completeness and vice-versa. You can see this by playing with the purity level using the --snquant option. Run Segment as shown above again with -P and see its default value. Then increase/decrease it for higher/lower purity and check the result as before. You will see that if you want the best purity, you have to sacrifice completeness and vice versa.

One interesting region to inspect in this image is the many bright peaks around the central parts of M51. Zoom into that region and inspect how many of them have actually been detected as true clumps. Do you have a good balance between completeness and purity? Also look out far into the wings of the group and inspect the completeness and purity there.

An easier way to inspect completeness (and only completeness) is to mask all the pixels detected as clumps and visually inspecting the rest of the pixels. You can do this using

$^{36}$ In AWK, $1$ refers to the first column, while in the shell script, it refers to the first argument.

$^{37}$ Some modifications are necessary based on the input dataset: depending on the dynamic range, you have to adjust the second and third arguments. But more importantly, depending on the dataset’s world coordinate system, you have to change the region width, in the AWK command. Otherwise the circle regions can be too small/large.
Arithmetic in a command like below. For easy reading of the command, we'll define the shell variable \( i \) for the image name and save the output in \texttt{masked.fits}.

\[
\begin{align*}
\$ \texttt{in} &= \texttt{r\_segmented.fits \ -hINPUT} \\
\$ \texttt{clumps} &= \texttt{r\_segmented.fits \ -hCLUMPS} \\
\$ \texttt{astarithmetic} \ \texttt{\$in} \ \texttt{\$clumps} \ 0 \ \texttt{gt} \ \texttt{nan} \ \texttt{where} \ \texttt{-oclumps\_masked.fits}
\end{align*}
\]

Inspecting \texttt{clumps\_masked.fits}, you can see some very diffuse peaks that have been missed, especially as you go farther away from the group center and into the diffuse wings. This is due to the fact that with this configuration, we have focused more on the sharper clumps. To put the focus more on diffuse clumps, you can use a wider convolution kernel. Using a larger kernel can also help in detecting the existing clumps to fainter levels (thus better separating them from the surrounding diffuse signal).

You can make any kernel easily using the \texttt{--kernel} option in Section 8.1 [MakeProfiles], page 440. But note that a larger kernel is also going to wash-out many of the sharp/small clumps close to the center of M51 and also some smaller peaks on the wings. Please continue playing with Segment’s configuration to obtain a more complete result (while keeping reasonable purity). We’ll finish the discussion on finding true clumps at this point.

The properties of the clumps within M51, or the background objects can then easily be measured using Section 7.4 [MakeCatalog], page 392. To measure the properties of the background objects (detected as clumps over the diffuse region), you shouldn’t mask the diffuse region. When measuring clump properties with Section 7.4 [MakeCatalog], page 392, and using the \texttt{--clumpscat}, the ambient flux (from the diffuse region) is calculated and subtracted. If the diffuse region is masked, its effect on the clump brightness cannot be calculated and subtracted.

To keep this tutorial short, we’ll stop here. See Section 2.2.13 [Segmentation and making a catalog], page 52, and Section 7.3 [Segment], page 381, for more on using Segment, producing catalogs with MakeCatalog and using those catalogs.

### 2.4 Building the extended PSF

Deriving the extended PSF of an image is very important in many aspects of the analysis of the objects within it. Gnuastro has a set of installed scripts, designed to simplify the process following the recipe of Infante-Sainz et al. (2020, \url{https://arxiv.org/abs/1911.01430}); for more, see Section 10.5 [PSF construction and subtraction], page 497. An overview of the process is given in Section 10.5.1 [Overview of the PSF scripts], page 498.

#### 2.4.1 Preparing input for extended PSF

We will use an image of the M51 galaxy group in the \texttt{r} (SDSS) band of the Javalambre Photometric Local Universe Survey (J-PLUS) to extract its extended PSF. For more information on J-PLUS, and its unique features visit: \url{http://www.j-plus.es}.

First, let’s download the image from the J-PLUS web page using \texttt{wget}. But to have a generalize-able, and easy to read command, we’ll define some base variables (in all-caps) first. After the download is complete, open the image with SAO DS9 (or any other FITS viewer you prefer!) to have a feeling of the data (and of course, enjoy the beauty of M51 in such a wide field of view):

\[
\$ \texttt{urlend=\texttt{\textasciitilde{jplus-dr2/get\_fits?id=67510}}}
\]
$ urlbase="http://archive.cefca.es/catalogues/vo/siap/"
$ mkdir jplus-dr2
$ wget $urlbase$urlend -O jplus-dr2/67510.fits.fz
$ astscript-fits-view jplus-dr2/67510.fits.fz

After enjoying the large field of view, have a closer look at the edges of the image. Please zoom in to the corners. You will see that on the edges, the pixel values are either zero or with significantly different values than the main body of the image. This is due to the dithering pattern that was used to make this image and happens in all imaging surveys. To avoid potential issues or problems that these regions may cause, we will first crop out the main body of the image with the command below. To keep the top-level directory clean, let’s also put the crop in a directory called flat.

$ mkdir flat
$ astcrop jplus-dr2/67510.fits.fz --section=225:9275,150:9350 --mode=img -oflat/67510.fits
$ astscript-fits-view flat/67510.fits

Please zoom into the edges again, you will see that they now have the same noise-level as the rest of the image (the problematic parts are now gone).

### 2.4.2 Saturated pixels and Segment’s clumps

A constant-depth (flat) image was created in the previous section (Section 2.4.1 [Preparing input for extended PSF], page 97). As explained in Section 10.5.1 [Overview of the PSF scripts], page 498, an important step when building the PSF is to mask other sources in the image. Therefore, before going onto selecting stars, let’s detect all significant signal, and identify the clumps of background objects over the wings of the extended PSF.

There is a problem however: the saturated pixels of the bright stars are going to cause problems in the segmentation phase. To see this problem, let’s make a $1000 \times 1000$ crop around a bright star to speed up the test (and its solution). Afterwards we’ll apply the solution to the whole image.

$ astcrop flat/67510.fits --mode=wcs --widthinpix --width=1000 --center=203.3916736,46.7968652 --output=saturated.fits
$ astnoisechisel saturated.fits --output=sat-nc.fits
$ astsegment sat-nc.fits --output=sat-seg.fits
$ astscript-fits-view sat-seg.fits

Have a look at the CLUMPS extension. You will see that instead of a single clump at the center of the bright star, we have many clumps! This has happened because of the saturated pixels! When saturation occurs, the sharp peak of the profile is lost (like cutting off the tip of a mountain to build a telescope!) and all saturated pixels get a noisy value close to the saturation level. To see this saturation noise run the last command again and in SAO DS9, set the “Scale” to “min max” and zoom into the center. You will see the noisy saturation pixels at the center of the star in red.

This noise-at-the-peak disrupts Segment’s assumption to expand clumps from a local maxima: each noisy peak is being treated as a separate local maxima and thus a separate clump. For more on how Segment defines clumps, see Section 3.2.1 and Figure 8 of

---

38 Recall the cropping in a previous tutorial for a similar reason (varying “depth” across the image): Section 2.2.4 [Dataset inspection and cropping], page 32.
Akhlaghi & Ichikawa 2015 (https://arxiv.org/abs/1505.01664). To have the center identified as a single clump, we should mask these saturated pixels in a way that suites Segment’s non-parametric methodology.

First we need to find the saturation level! The saturation level is usually fixed for any survey or input data that you receive from a certain database, so you will usually have to do this only once (the first time you get data from that database). Let’s make a smaller crop of 50 × 50 pixels around the star with the first command below. With the next command, please look at the crop with DS9 to visually understand the problem. You will see the saturated pixels as the noisy red pixels in the center of the image. A non-saturated star will have a single pixel as the maximum and won’t have a such a large area covered by a noisy constant value (find a few stars in the image and see for yourself). Visual and qualitative inspection of the process is very important for understanding the solution.

```
$ astcrop saturated.fits --mode=wcs --widthinpix --width=50 \\
   --center=203.3916736,46.7968652 --output=sat-center.fits
$ astscript-fits-view sat-center.fits --ds9scale=minmax
```

To quantitatively identify the saturation level in this image, let’s have a look at the distribution of pixels with a value larger than 100 (above the noise level):

```
$ aststatistics sat-center.fits --greaterequal=100
```

```
Histogram:
|*|*
|*|*
|*|*|*
|**|***|****
|*****|******|*******
|*********************|*************|***********
|-------------------------------
```

The peak you see in the right end (larger values) of the histogram shows the saturated pixels (a constant level, with some scatter due to the large Poisson noise). If there was no saturation, the number of pixels should have decreased at increasing values; until reaching the maximum value of the profile in one pixel. But that is not the case here. Please try this experiment on a non-saturated (fainter) star to see what we mean.

If you still haven’t experimented on a non-saturated star, please stop reading this tutorial! Please open flat/67510.fits in DS9, select a fainter/smaller star and repeat the last three commands (with a different center). After you have confirmed the point above (visually, and with the histogram), please continue with the rest of this tutorial.

Finding the saturation level is easy with Statistics (by using the `--lessthan` option until the histogram becomes as expected: only decreasing). First, let’s try `--lessthan=3000`:

```
$ aststatistics sat-center.fits --greaterequal=100 --lessthan=3000
```

```
```

Histogram:
We still see an increase in the histogram around 3000. Let’s try a threshold of 2500:

```
$ aststatistics sat-center.fits --greaterequal=100 --lessthan=2500
```

Histogram:

```
|*  |
|*  |
|** |
|***|
|****|
|*****|
|******|
*********
*************
*********************************
------------------------------------------------------------------
```

The peak at the large end of the histogram has gone! But let’s have a closer look at the values (the resolution of an ASCII histogram is limited!). To do this, we’ll ask Statistics to save the histogram into a table with the `--histogram` option, then look at the last 20 rows:

```
$ aststatistics sat-center.fits --greaterequal=100 --lessthan=2500 \
   --histogram --output=sat-center-hist.fits
$ asttable sat-center-hist.fits --tail=20
```

```
2021.1849112701 1
2045.0495397186 0
2068.9141681671 1
2092.7787966156 1
2116.6434250641 0
2140.5080535126 0
2164.3726819611 0
2188.2373104095 0
2212.101938858 1
2235.9665673065 1
2259.831195755 2
2283.6958242035 0
```
Since the number of points at the extreme end are increasing (from 1 to 2), we therefore see that a value 2500 is still above the saturation level (the number of pixels has started to increase)! A more reasonable saturation level for this image would be 2200! As an exercise, you can try automating this selection with AWK.

Therefore, we can set the saturation level in this image\(^{39}\) to be 2200. Let’s mask all such pixels with the command below:

```bash
$ astarithmetic saturated.fits set-i i i 2200 gt nan where \
   --output=sat-masked.fits
$ astscript-fits-view sat-masked.fits --ds9scale=minmax
```

You will see the peaks of several bright stars, not just the central very bright star. Zoom into each of the peaks you see. Besides the central very bright one that we were looking at closely until now, only one other star is saturated (its center is NaN, or Not-a-Number). Try to find it.

But we aren’t done yet! Please zoom-in to that central bright star and have another look on the edges of the vertical “bleeding” saturated pixels, there are strong positive/negative values touching it (almost like “waves”). These will also cause problems and have to be masked! So with a small addition to the previous command, let's dilate the saturated regions (with 2-connectivity, or 8-connected neighbors) four times and have another look:

```bash
$ astarithmetic saturated.fits set-i i i 2200 gt \
   2 dilate 2 dilate 2 dilate 2 dilate \
   nan where --output=sat-masked.fits
$ astscript-fits-view sat-masked.fits --ds9scale=minmax
```

Now that saturated pixels (and their problematic neighbors) have been masked, we can convolve the image (recall that Segment will use the convolved image for identifying clumps) with the command below. However, we will use the Spatial Domain convolution which can account for blank pixels (for more on the pros and cons of spatial and frequency domain convolution, see Section 6.3.3 [Spatial vs. Frequency domain], page 323). We will also create a Gaussian kernel with FWHM = 2 pixels, truncated at 5 × FWHM.

```bash
$ astmkprof --kernel=gaussian,2,5 --oversample=1 -okernel.fits
$ astconvolve sat-masked.fits --kernel=kernel.fits --domain=spatial \
   --output=sat-masked-conv.fits
$ astscript-fits-view sat-masked-conv.fits --ds9scale=minmax
```

Please zoom-in to the star and look closely to see how after spatial-domain convolution, the problematic pixels are still NaN. But Segment requires the profile to start with a maximum

---

\(^{39}\) In raw exposures, this value is usually around 65000 (close to \(2^{16}\), since most CCDs have 16-bit pixels; see Section 4.5 [Numeric data types], page 178). But that is not the case here, because this is a processed/stacked image that has been calibrated.
value and decrease. So before feeding into Segment, let’s fill the blank values with the maximum value of the neighboring pixels in both the input and convolved images (see Section 6.2.3.7 [Interpolation operators], page 287):

```
$ astarithmetic sat-masked.fits 2 interpolate-maxofregion \
   --output=sat-fill.fits

$ astarithmetic sat-masked-conv.fits 2 interpolate-maxofregion \
   --output=sat-fill-conv.fits

$ astscript-fits-view sat-fill* --ds9scale=minmax
```

Have a closer look at the opened images. Please zoom-in (you will notice that they are already matched and locked, so they will both zoom-in together). Go to the centers of the saturated stars and confirm how they are filled with the largest non-blank pixel. We can now feed this image to NoiseChisel and Segment as the convolved image:

```
$ astnoisechisel sat-fill.fits --convolved=sat-fill-conv.fits \
   --output=sat-nc.fits

$ astsegment sat-nc.fits --convolved=sat-fill-conv.fits \
   --output=sat-seg.fits --rawoutput

$ ds9 -mecube sat-seg.fits -zoom to fit -scale limits -1 1
```

See the CLUMPS extension. Do you see how the whole center of the star has indeed been identified as a single clump? We thus achieved our aim and didn’t let the saturated pixels harm the identification of the center!

If the issue was only clumps (like in a normal deep image processing), this was the end of Segment’s special considerations. However, in the scenario here, with the very extended wings of the bright stars, it usually happens that background objects become “clumps” in the outskirts and will rip the bright star outskirts into separate “objects”. In the next section (Section 2.4.3 [One object for the whole detection], page 102), we’ll describe how you can modify Segment to avoid this issue.

### 2.4.3 One object for the whole detection

In Section 2.4.2 [Saturated pixels and Segment’s clumps], page 98, we described how you can run Segment such that saturated pixels don’t interfere with its clumps. However, due to the very extended wings of the PSF, the default definition of “objects” should also be modified for the scenario here. To better see the problem, let’s inspect now the OBJECTS extension, focusing on those objects with a label between 50 to 150 (which include the main star):

```
$ astscript-fits-view sat-seg.fits -hOBJECTS --ds9scale="limits 50 150"
```

We can see that the detection corresponding to the star has been broken into different objects. This is not a good object segmentation image for our scenario here. Since those objects in the outer wings of the bright star’s detection harbor a lot of the extended PSF. We want to keep them with the same “object” label as the star (we only need to mask the “clumps” of the background sources). To do this, we’ll make the following changes to Segment’s options (see Section 7.3.1.2 [Segmentation options], page 387, for more on this options):

- Since we want the extended diffuse flux of the PSF to be taken as a single object, we want all the grown clumps to touch. Therefore, it is necessary to decrease \(-g\)thresh to very low values, like \(-10\). Recall that its value is in units of the input standard
deviation, so \(-gthresh=-10\) corresponds to \(-10\sigma\). The default value is not for such extended sources that dominate all background sources.

- Since we want all connected grown clumps to be counted as a single object in any case, we’ll set \(-objbordersn=0\) (its smallest possible value).

Let’s make these changes and check if the star has been kept as a single object in the \texttt{OBJECTS} extension or not:

\begin{verbatim}
$ astsegment sat-nc.fits --convolved=sat-fill-conv.fits \\
    --gthresh=-10 --objbordersn=0 \\
    --output=sat-seg.fits --rawoutput
$ astscript-fits-view sat-seg.fits -hOBJECTS --ds9scale="limits 50 150"
\end{verbatim}

Now we can extend these same steps to the whole image. To detect signal, we can run NoiseChisel using the command below. We modified the default value to two of the options, below you can see the reason for these changes. See Section 2.3 [Detecting large extended targets], page 76, for more on optimizing NoiseChisel.

- Since the image is so large, we have increased \(-interpnumngb\) to get better outlier statistics on the tiles. The default value is primarily for small images, so this is usually the first thing you should do when running NoiseChisel on a real/large image.
- Since the image is not too deep (made from few exposures), it doesn’t have strong correlated noise, so we’ll decrease \(-detgrowquant\) and increase \(-detgrowmaxholesize\) to better extract signal.

Furthermore, since both NoiseChisel and Segment need a convolved image, we’ll do the convolution before and feed it to both (to save running time). But in the first command below, let’s delete all the temporary files we made above.

\begin{verbatim}
Since the image is large (+300 MB), to avoid wasting storage, any temporary file that is no longer necessary for later processing is deleted after it is used. You can visually check each of them with DS9 before deleting them (or not delete them at all!). Generally, within a pipeline it is best to remove such large temporary files, because space runs out much faster than you think (for example once you get good results and want to use more fields).
\end{verbatim}

\begin{verbatim}
$ rm *.fits
$ mkdir label
$ astmkprof --kernel=gaussian,2,5 --oversample=1 \\
    -olabel/kernel.fits
$ astarithmetic flat/67510.fits set-i i i 2200 gt \\
    2 dilate 2 dilate 2 dilate 2 dilate nan where \\
    --output=label/67510-masked-sat.fits
$ astconvolve label/67510-masked-sat.fits --kernel=label/kernel.fits \\
    --domain=spatial --output=label/67510-masked-conv.fits
$ rm label/kernel.fits
$ astarithmetic label/67510-masked-sat.fits 2 interpolate-maxofregion \\
    --output=label/67510-fill.fits
$ astarithmetic label/67510-masked-conv.fits 2 interpolate-maxofregion \\
    --output=label/67510-fill-conv.fits
$ rm label/67510-masked-conv.fits
$ astnoisechisel label/67510-fill.fits --interpnumngb=100 \ 
\end{verbatim}
--detgrowquant=0.8 --detgrowmaxholesize=100000 \ 
--convolved=label/67510-fill-conv.fits \ 
--output=label/67510-nc.fits
$ rm label/67510-fill.fits
$ astsegment label/67510-nc.fits --output=label/67510-seg-raw.fits \ 
--convolved=label/67510-fill-conv.fits --rawoutput \ 
--gthresh=-10 --objbordersn=0
$ rm label/67510-fill-conv.fits
$ astscript-fits-view label/67510-seg-raw.fits

We see that the saturated pixels haven’t caused any problem and the central clumps/objects of bright stars are now a single clump/object. We can now proceed to estimating the outer PSF. But before that, let’s make a “standard” segment output: one that can safely be fed into MakeCatalog for measurements and can contain all necessary outputs of this whole process in a single file (as multiple extensions).

The main problem is again the saturated pixels: we interpolated them to be the maximum of their nearby pixels. But this will cause problems in any measurement that is done over those regions. To let MakeCatalog know that those pixels shouldn’t be used, the first extension of the file given to MakeCatalog should have blank values on those pixels. We will do this with the commands below:

## First HDU of Segment (Sky-subtracted input)
$ astarithmetic label/67510-nc.fits -hINPUT-NO-SKY \ 
label/67510-masked-sat.fits isblank nan where \ 
--output=label/67510-seg.fits
$ astfits label/67510-seg.fits --update=EXTNAME,INPUT-NO-SKY

## Second and third HDUs: CLUMPS and OBJECTS
$ astfits label/67510-seg-raw.fits --copy=CLUMPS --copy=OBJECTS \ 
--output=label/67510-seg.fits

## Fourth HDU: Sky standard deviation (from NoiseChisel):
$ astfits label/67510-nc.fits --copy=SKY_STD \ 
--output=label/67510-seg.fits

## Clean up all the un-necessary files:
$ rm label/67510-masked-sat.fits label/67510-nc.fits \ 
label/67510-seg-raw.fits

You can now simply run MakeCatalog on this image and be sure that saturated pixels won’t affect the measurements. As one example, you can use MakeCatalog to find the clumps containing saturated pixels: recall that the --area column only calculates the area of non-blank pixels, while --geoarea calculates the area of the label (independent of their blank-ness in the values image):

$ astmkcatalog label/67510-seg.fits --ids --ra --dec --area --geoarea \ 
--clumpscat --output=cat.fits

The information of the clumps that have been affected by saturation can easily be found by selecting those with a differing value in the AREA and AREA_FULL columns:
$ asttable cat.fits -hCLUMPS | awk '$5!=$6'
$ asttable cat.fits -hCLUMPS | awk '$5!=$6' | wc -l

## Using Table arithmetic (compared to AWK, you can use column names, save as FITS, and be faster):
$ asttable cat.fits -hCLUMPS -cRA,DEC --noblankend=3 \
   -c'arith AREA AREA AREA_FULL eq nan where'

## Remove the table (which was just for a demo)
$ rm cat.fits

We are now ready to start building the outer parts of the PSF in Section 2.4.4 [Building outer part of PSF], page 105.

### 2.4.4 Building outer part of PSF

In Section 2.4.1 [Preparing input for extended PSF], page 97, we described how to create a Segment clump and object map, while accounting for saturated stars and not having over-fragmentation of objects in the outskirts of stars. We are now ready to start building the extended PSF.

First we will build the outer parts of the PSF, so we want the brightest stars. You will see we have several bright stars in this very large field of view, but we don’t yet have a feeling how many they are, and at what magnitudes. So let’s use Gnuastro’s Query program to find the magnitudes of the brightest stars (those brighter than g-magnitude 10 in Gaia early data release 3, or eDR3). For more on Query, see Section 5.4 [Query], page 249.

```
$ astquery gaia --dataset=edr3 --overlapwith=flat/67510.fits \
   --range=phot_g_mean_mag,-inf,10 \
   --output=flat/67510-bright.fits
```

Now, we can easily visualize the magnitude and positions of these stars using `astscript-ds9-region` and the command below (for more on this script, see Section 10.3 [SAO DS9 region files from table], page 492)

```
$ astscript-ds9-region flat/67510-bright.fits -cra,dec \
   --namecol=phot_g_mean_mag \
   --command="ds9 flat/67510.fits -zoom to fit -zscale"
```

You can see that we have several stars between magnitudes 6 to 10. Let’s use `astscript-psf-select-stars` in the command below to select the relevant stars in the image (the brightest; with a magnitude between 6 to 10). The advantage of using this script (instead of a simple `--range` in Table), is that it will also check distances to nearby stars and reject those that are too close (and not good for constructing the PSF). Since we have very bright stars in this very wide-field image, we will also increase the distance to nearby neighbors with brighter or similar magnitudes (the default value is 1 arcmin). To do this, we will set `--mindistdeg=0.02`, which corresponds to 1.2 arcmin.

```
$ mkdir outer
$ astscript-psf-select-stars flat/67510.fits \ 
   --magnituderange=6,10 --mindistdeg=0.02 \ 
   --output=outer/67510-6-10.fits
```
Let’s have a look at the selected stars in the image (it is very important to visually check every step when you are first discovering a new dataset).

```
$ astscript-ds9-region outer/67510-6-10.fits -cra,dec \\
   --namecol=phot_g_mean_mag \\
   --command="ds9 flat/67510.fits -zoom to fit -zscale"
```

Now that the catalog of good stars is ready, it is time to construct the individual stamps from the catalog above. To do that, we’ll use `astscript-psf-stamp`. One of the most important parameters for this script is the normalization radii `--normradii`. This parameter defines a ring for the flux normalization of each star stamp. The normalization of the flux is necessary because each star has a different brightness, and consequently, it is crucial for having all the stamps with the same flux level in the same region. Otherwise the final stack of the different stamps would have no sense. Depending on the PSF shape, internal reflections, ghosts, saturated pixels, and other systematics, it would be necessary to choose the `--normradii` appropriately.

The selection of the normalization radii is something that requires a good understanding of the data. To do that, let’s use two useful parameters that will help us in the checking of the data: `--tmpdir` and `--keeptmp`:

- With `--tmpdir=checking-normradii` all temporary files, including the radial profiles, will be save in that directory (instead of an internally-created name).
- With `--keeptmp` we won’t remove the temporal files, so it is possible to have a look at them (by default the temporary directory gets deleted at the end). It is necessary to specify the `--normradii` even if we don’t know yet the final values. Otherwise the script will not generate the radial profile.

As a consequence, in this step we put the normalization radii equal to the size of the stamps. By doing this, the script will generate the radial profile of the entire stamp. In this particular step we set it to `--normradii=500,510`. Furthermore, since there are several stars, we iterate over each row of the catalog using a while loop.

```
$ counter=1
$ mkdir finding-normradii
$ asttable outer/67510-6-10.fits \
   | while read -r ra dec mag; do
      astscript-psf-stamp label/67510-seg.fits \
         --mode=wcs \
         --stampwidth=1000 \
         --center=$ra,$dec \
         --normradii=500,510 \
         --segment=label/67510-seg.fits \
         --output=finding-normradii/$counter.fits \
         --tmpdir=finding-normradii --keeptmp; \
      counter=$((counter+1)); \
   done
```

First let’s have a look at all the masked postage stamps of the cropped stars. Once they all open, feel free to zoom-in, they are all matched and locked. It is always good to check the different stamps to ensure the quality and possible two dimensional features that are difficult to detect from the radial profiles (i.e., ghosts, internal reflections, etc.).
$ astscript-fits-view finding-normradii/cropped-masked*.fits

If everything looks good in the image, let’s open all the radial profiles and visually check those with the command below. Note that astscript-fits-view calls the topcat graphic user interface (GUI) program to visually inspect (plot) tables. If you don’t already have it, see Section B.2 [TOPCAT], page 703.

$ astscript-fits-view finding-normradii/rprofile*.fits

After some study of this data, we could say that a good normalization ring is those pixels between R=20 and R=30 pixels. Such a ring ensures having a high number of pixels so the estimation of the flux normalization will be robust. Also, at such distance from the center the signal to noise is high and there are not obvious features that can affect the normalization. Note that the profiles are different because we are considering a wide range of magnitudes, so the fainter stars are much more noisy. However, in this tutorial we will keep these stars in order to have a higher number of stars for the outer part. In a real case scenario, we should look for stars with a much more similar brightness (smaller range of magnitudes) to not lose signal to noise as a consequence of the inclusion of fainter stars.

$ rm -r finding-normradii
$ counter=1
$ mkdir outer/stamps
$ asttable outer/67510-6-10.fits
   | while read -r ra dec mag; do
      $ astscript-psf-stamp label/67510-seg.fits
         --mode=wcs
         --stampwidth=1000
         --center=$ra,$dec
         --normradii=20,30
         --segment=label/67510-seg.fits
         --output=outer/stamps/67510-$counter.fits;
      counter=$((counter+1));
   done

After the stamps are created, we need to stack them together with a simple Arithmetic command (see Section 6.2.3.5 [Stacking operators], page 283). The stack is done using the sigma-clipped mean operator that will preserve more of the signal, while rejecting outliers (more than $3\sigma$ with a tolerance of 0.2, for more on sigma-clipping see Section 7.1.3 [Sigma clipping], page 343). Just recall that we need to specify the number of inputs into the stacking operators, so we are reading the list of images and counting them as separate variables before calling Arithmetic.

$ imgs=outer/stamps/*.fits
$ numimgs=$(echo $imgs | wc -w)
$ astarithmetic $imgs $numimgs 3 0.2 sigclip-mean -g1
   --output=outer/stack.fits --wcsfile=none

Did you notice the --wcsfile=none option above? With it, the stacked image no longer has any WCS information. This is natural, because the stacked image doesn’t correspond to any specific region of the sky any more.
Let’s compare this stacked PSF with the images of the individual stars that were used to create it. You can clearly see that the number of masked pixels is significantly decreased and the PSF is much more “cleaner”.

```
$ astscript-fits-view outer/stack.fits outer/stamps/*.fits
```

However, the saturation in the center still remains. Also, because we didn’t have too many images, some regions still are very noisy. If we had more bright stars in our selected magnitude range, we could have filled those outer remaining patches. In a large survey like J-PLUS (that we are using here), you can simply look into other fields that were observed soon before/after the image ID 67510 that we used here (to have a similar PSF) and get more stars in those images to add to these. In fact, the J-PLUS DR2 image ID of the field above was intentionally preserved during the steps above to show how easy it is to use images from other fields and blend them all into the output PSF.

### 2.4.5 Inner part of the PSF

In Section 2.4.4 [Building outer part of PSF], page 105, we were able to create a stack of the outer-most behavior of the PSF in a J-PLUS survey image. But the central part that was affected by saturation and non-linearity is still remaining, and we still don’t have a “complete” PSF! In this section, we will use the same steps before to make stacks of more inner regions of the PSF to ultimately unite them all into a single PSF in Section 2.4.6 [Uniting the different PSF components], page 109.

For the outer PSF, we selected stars in the magnitude range of 6 to 10. So let’s have a look and see how many stars we have in the magnitude range of 12 to 13 with a more relaxed condition on the minimum distance for neighbors, `--mindistdeg=0.01` (36 arcsec, since these stars are fainter), and use the ds9 region script to visually inspect them:

```
$ mkdir inner
$ astscript-psf-select-stars flat/67510.fits \
   --magnituderange=12,13 --mindistdeg=0.01 \
   --output=inner/67510-12-13.fits

$ astscript-ds9-region inner/67510-12-13.fits -cra,dec \
   --namecol=phot_g_mean_mag \
   --command="ds9 flat/67510.fits -zoom to fit -zscale"
```

We have 41 stars, but if you zoom into their centers, you will see that they don’t have any major bleeding-vertical saturation any more. Only the very central core of some of the stars is saturated. We can therefore use these stars to fill the strong bleeding footprints that were present in the outer stack of `outer/stack.fits`. Similar to before, let’s build ready-to-stack crops of these stars. To get a better feeling of the normalization radii, follow the same steps of Section 2.4.4 [Building outer part of PSF], page 105, (setting `--tmpdir` and `--keeptmp`). In this case, since the stars are fainter, we can set a smaller size for the individual stamps, `--stampwidth=500`, to speed up the calculations:

```
$ counter=1
$ mkdir inner/stamps
$ asttable inner/67510-12-13.fits \ 
   while read -r ra dec mag; do \
   astscript-psf-stamp label/67510-seg.fits \
   ```
We are now ready to unite the two stacks we have constructed: the outer and the inner parts.

### 2.4.6 Uniting the different PSF components

In Section 2.4.4 [Building outer part of PSF], page 105, we built the outer part of the extended PSF and the inner part was built in Section 2.4.5 [Inner part of the PSF], page 108. The outer part was built with very bright stars, and the inner part using fainter stars to not have saturation in the core of the PSF. The next step is to join these two parts in order to have a single PSF. First of all, let’s have a look at the two stacks and also to their radial profiles to have a good feeling of the task. Note that you will need to have TOPCAT to run the last command and plot the radial profile (see Section B.2 [TOPCAT], page 703).

```
$ astscript-fits-view outer/stack.fits inner/stack.fits
$ astscript-radial-profile outer/stack.fits -o outer/profile.fits
$ astscript-radial-profile inner/stack.fits -o inner/profile.fits
$ astscript-fits-view outer/profile.fits inner/profile.fits
```

From the visual inspection of the images and the radial profiles, it is clear that we have saturation in the center for the outer part. Note that the absolute flux values of the PSFs are meaningless since they depend on the normalization radii we used to obtain them. The uniting step consists in scaling up (or down) the inner part of the PSF to have the same flux at at the junction radius, and then, use that flux-scaled inner part to fill the center of the outer PSF. To get a feeling of the process, first, let’s open the two radial profiles and find the factor manually first:

1. Run this command to open the two tables in Section B.2 [TOPCAT], page 703:
   ```
   $ astscript-fits-view outer/profile.fits inner/profile.fits
   ```
2. On the left side of the screen, under “Table List”, you will see the two imported tables. Click on the first one (profile of the outer part) so it is shown first.
3. Under the “Graphics” menu item, click on “Plane plot”. A new window will open with the plot of the first two columns: RADIUS on the horizontal axis and MEAN on the vertical. The rest of the steps are done in this window.
4. In the bottom settings, within the left panel, click on the “Axes” item. This will allow customization of the plot axises.
5. In the bottom-right panel, click on the box in front of “Y Log” to make the vertical axis logarithmic-scaled.

6. On the “Layers” menu, select “Add Position Control” to allow adding the profile of the inner region. After it, you will see that a new red-blue scatter plot icon opened on the bottom-left menu (with a title of <no table>).

7. On the bottom-right panel, in the drop-down menu in front of Table:, select 2: profile.fits. Afterwards, you will see the radial profile of the inner stack as the newly added blue plot. Our goal here is to find the factor that is necessary to multiply with the inner profile so it matches the outer one.

8. On the bottom-right panel, in front of Y:, you will see MEAN. Click in the white-space after it, and type this: \*100. This will display the MEAN column of the inner profile, after multiplying it by 100. Afterwards, you will see that the inner profile (blue) matches more cleanly with the outer (red); especially in the smaller radii. At larger radii, it doesn’t drop like the red plot. This is because of the extremely low signal-to-noise ratio at those regions in the fainter stars used to make this stack.

9. Take your mouse cursor over the profile, in particular over the bump around a radius of 100 pixels. Scroll your mouse down-ward to zoom-in to the profile and up-ward to zoom-out. You can click-and-hold any part of the profile and if you move your cursor (while still holding the mouse-button) to look at different parts of the profile. This is particular helpful when you have zoomed-in to the profile.

10. Zoom-in to the bump around a radius of 100 pixels until the horizontal axis range becomes around 50 to 130 pixels.

11. You clearly see that the inner stack (blue) is much more noisy than the outer (red) stack. By “noisy”, we mean that the scatter of the points is much larger. If you further zoom-out, you will see that the shallow slope at the larger radii of the inner (blue) profile has also affected the height of this bump in the inner profile. This is a very important point: this clearly shows that the inner profile is too noisy at these radii.

12. Click-and-hold your mouse to see the inner parts of the two profiles (in the range 0 to 80). You will see that for radii less than 40 pixels, the inner profile (blue) points loose their scatter (and thus have a good signal-to-noise ratio).

13. Zoom-in to the plot and follow the profiles until smaller radii (for example 10 pixels). You see that for each radius, the inner (blue) points are consistently above the outer (red) points. This shows that the \*100 factor we selected above was too much.

14. In the bottom-right panel, change the 100 to 80 and zoom-in to the same region. At each radius, the blue points are now below the red points, so the scale-factor 80 is not enough. So let’s increase it and try 90. After zooming-in, you will notice that in the inner-radii (less than 30 pixels), they are now very similar. The ultimate aim of the steps below is to find this factor automatically.

15. But before continuing, let’s focus on another important point about the central regions: non-linearity and saturation. While you are zoomed-in (from the step above), follow (click-and-drag) the profile towards smaller radii. You will see that smaller than a radius of 10, they start to diverge. But this time, the outer (red) profile is getting a shallower slope and diverges significantly from about the radius of 8. We had masked all saturated pixels before, so this divergence for radii smaller than 10 shows the effect of the CCD’s non-linearity (where the number of electrons will not be linearly correlated.
with the number of incident photons). This is present in all CCDs and pixels beyond this level should not be used in measurements (or properly corrected).

The items above were only listed so you get a good mental/visual understanding of the logic behind the operation of the next script (and to learn how to tune its parameters where necessary): `astscript-psf-scale-factor`. This script is more general than this particular problem, but can be used for this special case also. Its job is to take a model of an object (PSF, or inner stack in this case) and the position of an instance of that model (a star, or the outer stack in this case) in a larger image.

Instead of dealing with radial profiles (that enforce a certain shape), this script will put the centers of the inner and outer PSFs over each other and divide the outer by the inner. Let’s have a look with the command below. Just note that we are running it with `--keeptmp` so the temporary directory with all the intermediate files remain for further clarification:

```
$ astscript-psf-scale-factor outer/stack.fits \
   --psf=inner/stack.fits --center=501,501 \
   --mode=img --normradii=10,15 --keeptmp
$ astscript-fits-view stack_psfmodelscalefactor/cropped-*.fits \
   stack_psfmodelscalefactor/for-factor-*.fits
```

With the second command, you see the four steps of the process: the first two images show the cropped outer and inner stacks (to same width image). The third shows the radial position of each pixel (which is used to only keep the pixels within the desired radial range). The fourth shows the per-pixel division of the outer by the inner within the requested radii. The sigma-clipped median of these pixels is finally reported. Unlike the radial profile method (which averages over a circular/elliptical annulus for each radius), this method imposes no a-priori shape on the PSF. This makes it very useful for complex PSFs (like the case here).

To continue, let’s remove the temporary directory and re-run the script but with `--quiet` mode so we can put the output in a shell variable.

```
$ rm -r stack_psfmodelscalefactor
$ scale=$(astscript-psf-scale-factor outer/stack.fits \
   --psf=inner/stack.fits --center=501,501 \
   --mode=img --normradii=10,15 --quiet)
$ echo $scale
```

Now that we know the scaling factor, we are ready to unite the outer and the inner part of the PSF. To do that, we will use the script `astscript-psf-unite` with the command below (for more on this script, see Section 10.5.4 [Invoking astscript-psf-unite], page 506). The basic parameters are the inner part of the PSF (given to `--inner`), the inner part’s scale factor (`--scale`), and the junction radius (`--radius`). The inner part is first scaled, and all the pixels of the outer image within the given radius are replaced with the pixels of the inner image. Since the flux factor was computed for a ring of pixels between 10 and 15 pixels, let’s set the junction radius to be 12 pixels (roughly in between 10 and 15):

```
$ astscript-psf-unite outer/stack.fits \
   --inner=inner/stack.fits --radius=12 \
   --scale=$scale --output=psf.fits
```

Let’s have a look at the outer stack and the final PSF with the command below. Since we want several other DS9 settings to help you directly see the main point, we are using `--ds9extra`. After DS9 is opened, you can see that the center of the PSF has now been
nicely filled. You can click on the “Edit” button and then the “Colorbar” and hold your cursor over the image and move it. You can see that besides filling the inner regions nicely, there is also no major discontinuity in the 2D image around the union radius of 12 pixels around the center.

```
$ astscript-fits-view outer/stack.fits psf.fits --ds9scale=minmax \
   --ds9extra="-scale limits 0 22000 -match scale" \
   --ds9extra="-lock scale yes -zoom 4 -scale log"
```

Nothing demonstrates the effect of a bad analysis than actually seeing a bad result! So let’s choose a bad normalization radial range (50 to 60 pixels) and unite the inner and outer parts based on that. The last command will open the two PSFs together in DS9, you should be able to immediately see the discontinuity in the union radius.

```
$ scale=$(astscript-psf-scale-factor outer/stack.fits \
   --psf=inner/stack.fits --center=501,501 \
   --mode=img --normradii=50,60 --quiet)

$ astscript-psf-unite outer/stack.fits \
   --inner=inner/stack.fits --radius=55 \
   --scale=$scale --output=psf-bad.fits

$ astscript-fits-view psf-bad.fits psf.fits --ds9scale=minmax \
   --ds9extra="-scale limits 0 50 -match scale" \
   --ds9extra="-lock scale yes -zoom 4 -scale log"
```

As you see, the selection of the normalization radii and the unite radius are very important. The first time you are trying to build the PSF of a new dataset, it has to be explored with a visual inspection of the images and radial profiles. Once you have found a good normalization radius for a certain part of the PSF in a survey, you can generally use it comfortably without change. But for a new survey, or a different part of the PSF, be sure to repeat the visual checks above to choose the best radii. As a summary, a good junction radius is one that:

- Is large enough to not let saturation and non-linearity (from the outer profile) into the inner region.
- Is small enough to have a sufficiently high signal to noise ratio (from the inner profile) to avoid adding noise in the union radius.

Now that the complete PSF has been obtained, let’s remove that bad-looking PSF, and stick with the nice and clean PSF for the next step in Section 2.4.7 [Subtracting the PSF], page 112.

```
$ rm -rf psf-bad.fits
```

### 2.4.7 Subtracting the PSF

Previously (in Section 2.4.6 [Uniting the different PSF components], page 109) we constructed a full PSF, from the central pixel to a radius of 500 pixels. Now, let’s use the PSF to subtract the scattered light from each individual star in the image.

By construction, the pixel values of the PSF came from the normalization of the individual stamps (that were created for stars of different magnitudes). As a consequence, it is
necessary to compute a scale factor to fit that PSF image to each star. This is done with
the same `astscript-psf-scale-factor` command that we used previously in Section 2.4.6
[Uniting the different PSF components], page 109. The difference is that now we are not
aiming to join two different PSF parts but looking for the necessary scale factor to match
the star with the PSF. Afterwards, we will use `astscript-psf-subtract` for placing the
PSF image at the desired coordinates within the same pixel grid as the image. Finally, once
the stars have been modeled by the PSF, we will subtract it.

First, let’s start with a single star. Later, when the basic idea has been explained, we
will generalize the method for any number of stars. With the following command we obtain
the coordinates (RA and DEC) and magnitude of the brightest star in the image (which is
on the top edge of the image):

```
$ mkdir single-star
$ center=$(asttable flat/67510-bright.fits --sort phot_g_mean_mag |
          --column=ra,dec --head 1 |
          awk '{printf "%s,%s", $1, $2}')
$ echo $center
```

With the center position of that star, let’s obtain the flux factor using the same normal-
ization ring we used for the creation of the outer part of the PSF:

```
$ scale=$(astscript-psf-scale-factor label/67510-seg.fits |
        --mode=wcs --quiet |
        --psf=psf.fits |
        --center=$center |
        --normradii=10,15 |
        --segment=label/67510-seg.fits)
```

Now we have all the information necessary to model the star using the PSF: the position
on the sky and the flux factor. Let’s use this data with the script `astscript-psf-subtract`
for modeling this star and have a look with DS9.

```
$ astscript-psf-subtract label/67510-seg.fits |
    --mode=wcs |
    --psf=psf.fits |
    --scale=$scale |
    --center=$center |
    --output=single-star/subtracted.fits
```

```
$ astscript-fits-view label/67510-seg.fits single-star/subtracted.fits |
    --ds9center=$center --ds9mode=wcs --ds9extra="-zoom 4"
```

You will notice that there is something wrong with this “subtraction”! The box of the
extended PSF is clearly visible! The sky noise under the box is clearly larger than the rest
of the noise in the image. Before reading on, please try to think about the cause of this
yourself.

To understand the cause, let’s look at the scale factor, the number of stamps used to
build the outer part (and its square root):

```
$ echo $scale
$ ls outer/stamps/*.fits | wc -l
$ ls outer/stamps/*.fits | wc -l | awk '{printf sqrt($1)}'
```
You see that the scale is almost 19! As a result, the PSF has been multiplied by 19 before being subtracted. However, the outer part of the PSF was created with only a handful of star stamps. When you stack $N$ images, the stack’s signal-to-noise ratio (S/N) improves by $\sqrt{N}$. We had 8 images for the outer part, so the S/N has only improved by a factor of just under 3! When we multiply the final stacked PSF with 19, we are also scaling up the noise by that same factor (most importantly: in the outer most regions where there is almost no signal). So the stacked image’s noise-level is $19/3 = 6.3$ times larger than the noise of the input image. This terrible noise-level is what you clearly see as the footprint of the PSF.

To confirm this, let’s use the commands below to subtract the faintest of the bright-stars catalog (note the use of --tail when finding the central position). You will notice that the scale factor ($\sim 1.3$) is now smaller than 3. So when we multiply the PSF with this factor, the PSF’s noise level is lower than our input image and we shouldn’t see any footprint like before. Note also that we are using a larger zoom factor, because this star is smaller in the image.

```
$ center=$(asttable flat/67510-bright.fits --sort phot_g_mean_mag --column=ra,dec --tail 1 | awk '{printf "%s,%s", $1, $2}'}

$ scale=$(astscript-psf-scale-factor label/67510-seg.fits --mode=wcs --quiet --psf=psf.fits --center=$center --normradii=10,15 --segment=label/67510-seg.fits)

$ echo $scale

$ astscript-psf-subtract label/67510-seg.fits --mode=wcs --psf=psf.fits --scale=$scale --center=$center --output=single-star/subtracted.fits

$ astscript-fits-view label/67510-seg.fits single-star/subtracted.fits --ds9center=$center --ds9mode=wcs --ds9extra=-zoom 10
```

In a large survey like J-PLUS, its easy to use more and more bright stars from different pointings (ideally with similar FWHM and similar telescope properties) to improve the S/N of the PSF. As explained before, we designed the output files of this tutorial with the 67510 (which is this image’s pointing label in J-PLUS) where necessary so you see how easy it is to add more pointings to use in the creation of the PSF.

Let’s consider now more than one single star. We should have two things in mind:

- The brightest (subtract-able, see the point below) star should be the first star to be subtracted. This is because of its extended wings which may affect the scale factor of

---

40 For example in J-PLUS, the baffle of the secondary mirror was adjusted in 2017 because it produced extra spikes in the PSF. So all images after that date have a PSF with 4 spikes (like this one), while those before it have many more spikes.
nearby stars. So we should sort the catalog by brightness and come down from the brightest.

- We should only subtract stars where the scale factor is less than the S/N of the PSF (in relation to the data).

Since it can get a little complex, it's easier to implement this step as a script (that is heavily commented for you to easily understand every step; especially if you put it in a good text editor with color-coding!). You will notice that the script also creates a .log file, which shows which star was subtracted and which one wasn't (this is important, and will be used below!).

```bash
#!/bin/bash

# Abort the script on first error.
set -e

# ID of image to subtract stars from.
imageid=67510

# Get S/N level of the final PSF in relation to the actual data:
snlevel=$(ls outer/stamps/*.fits | wc -l | awk '{print sqrt($1)}')

# Put a copy of the image we want to subtract the PSF from in the final file (this will be over-written after each subtraction).
subtracted=subtracted/$imageid.fits
cp label/$imageid-seg.fits $subtracted

# Name of log-file to keep status of the subtraction of each star.
logname=subtracted/$imageid.log
echo "# Column 1: RA [deg, f64] Right ascension of star." > $logname
echo "# Column 2: Dec [deg, f64] Declination of star." >> $logname
echo "# Column 3: Stat [deg, f64] Status (1: subtracted)" >> $logname

# Go over each item in the bright star catalog:
asttable flat/67510-bright.fits -cra,dec --sort phot_g_mean_mag
while read -r ra dec; do
    center=$(echo $ra $dec | awk '{printf "%s,%s", $1, $2}')
    scale=$(astscript-psf-scale-factor label/67510-seg.fits
        --mode=wcs --quiet
        --psf=psf.fits
        --center=center
        --normradii=10,15
        --segment=label/67510-seg.fits)
```

# Subtract this star if the scale factor is less than the S/N level calculated above.
check=$(echo $snlevel $scale | awk '{if($1>$2) c="good"; else c="bad"; print c}')
if [ $check = good ]; then

    # A temporary file to subtract this star.
subtmp=subtracted/$imageid-tmp.fits

    # Subtract this star from the image where all previous stars were subtracted.
    astscript-psf-subtract $subtracted
        --mode=wcs
        --psf=psf.fits
        --scale=$scale
        --center=$center
        --output=$subtmp

    # Rename the temporary subtracted file to the final one:
    mv $subtmp $subtracted

    # Keep the status for this star.
    status=1
else

    # Let the user know this star didn't work, and keep the status for this star.
    echo "$center: $scale is larger than $snlevel"
    status=0
fi

    # Keep the status in a log file.
echo "$ra $dec $status" >> $logname
done

Copy the contents above into a file called subtract-psf-from-cat.sh and run the following commands. Just note that in the script above, we assumed the output is written in the subtracted/, directory, so we'll first make that.

$ mkdir subtracted
$ chmod +x subtract-psf-from-cat.sh
$ ./subtract-psf-from-cat.sh

$ astscript-fits-view label/67510-seg.fits subtracted/67510.fits

Can you visually find the stars that have been subtracted? Its a little hard, isn't it? This shows that you done a good job this time (the sky-noise is not significantly affected)! So let's subtract the actual image from the PSF-subtracted image to see the scattered light field of the subtracted stars. With the second command below we'll zoom into the brightest subtracted star, but of course feel free to zoom-out and inspect the others also.
$ astarithmetic label/67510-seg.fits subtracted/67510.fits - \
--output=scattered-light.fits -g1

$ center=$(asttable subtracted/67510.log --equal=Stat,1 --head=1 \
-cra,dec | awk '{printf "%s,%s", $1, $2}')

$ astscript-fits-view label/67510-seg.fits subtracted/67510.fits \
scattered-light.fits \
--ds9center=$center --ds9mode=wcs \
--ds9extra="-scale limits -0.5 1.5 -match scale" \
--ds9extra="-lock scale yes -zoom 10" \
--ds9extra="-tile mode column"

## We can always make it easily, so let's remove this.
$ rm scattered-light.fits

You will probably have noticed that in the scattered light field there are some patches 
that correspond to the saturation of the stars. Since we obtained the scattered light field 
by subtracting PSF-subtracted image from the original image, it is natural that we have 
such saturated regions. To solve such inconvenience, this script also has an option to not 
make the subtraction of the PSF but to give as output the modeled star. For doing that, 
it is necessary to run the script with the option --modelonly. We encourage the reader to 
obtain such scattered light field model. In some scenarios it could be interesting having such 
way of correcting the PSF. For example, if there are many faint stars that can be modeled 
at the same time because their flux don’t affect each other. In such situation, the task could 
be easily parallelized without having to wait to model the brighter stars before the fainter 
one. At the end, once all stars have been modeled, a simple Arithmetic command could 
be used to sum the different modeled-PSF stamps to obtain the entire scattered light field.

In general you see that the subtraction has been done nicely and almost all the extended 
wings of the PSF have been subtracted. The central regions of the stars aren’t perfectly 
subtracted:

- Some may get too dark at the center. This may be due to the non-linearity of the 
  CCD counting (as discussed previously in Section 2.4.6 [Uniting the different PSF 
  components], page 109).
- Others may have a strong gradient: one side is too positive and one side is too negative 
  (only in the very central few pixels). This is due to the non-accurate positioning: most 
  probably this happens because of imperfect astrometry.

Note also that during this process we assumed that the PSF doesn’t vary with the CCD 
position or any other parameter. In other words, we are obtaining an averaged PSF model 
from a few star stamps that are naturally different, and this also explains the residuals on 
each subtracted star.

We let as an interesting exercise the modeling and subtraction of other stars, for example, 
the non saturated stars of the image. By doing this, you will notice that in the core region 
the residuals are different compared to the residuals of brighter stars that we have obtained.
In general, in this tutorial we have showed how to deal with the most important challenges for constructing an extended PSF. Each image or dataset will have its own particularities that you will have to take into account when constructing the PSF.
3 Installation

The latest released version of Gnuastro source code is always available at the following URL:

http://ftpmirror.gnu.org/gnuastro/gnuastro-latest.tar.gz

Section 1.1 [Quick start], page 1, describes the commands necessary to configure, build, and install Gnuastro on your system. This chapter will be useful in cases where the simple procedure above is not sufficient, for example your system lacks a mandatory/optional dependency (in other words, you can’t pass the $ ./configure step), or you want greater customization, or you want to build and install Gnuastro from other random points in its history, or you want a higher level of control on the installation. Thus if you were happy with downloading the tarball and following Section 1.1 [Quick start], page 1, then you can safely ignore this chapter and come back to it in the future if you need more customization.

Section 3.1 [Dependencies], page 119, describes the mandatory, optional and bootstrapping dependencies of Gnuastro. Only the first group are required/mandatory when you are building Gnuastro using a tarball (see Section 3.2.1 [Release tarball], page 131), they are very basic and low-level tools used in most astronomical software, so you might already have them installed, if not they are very easy to install as described for each. Section 3.2 [Downloading the source], page 131, discusses the two methods you can obtain the source code: as a tarball (a significant snapshot in Gnuastro’s history), or the full history\(^1\). The latter allows you to build Gnuastro at any random point in its history (for example to get bug fixes or new features that are not released as a tarball yet).

The building and installation of Gnuastro is heavily customizable, to learn more about them, see Section 3.3 [Build and install], page 136. This section is essentially a thorough explanation of the steps in Section 1.1 [Quick start], page 1. It discusses ways you can influence the building and installation. If you encounter any problems in the installation process, it is probably already explained in Section 3.3.5 [Known issues], page 149. In Appendix B [Other useful software], page 702, the installation and usage of some other free software that are not directly required by Gnuastro but might be useful in conjunction with it is discussed.

3.1 Dependencies

A minimal set of dependencies are mandatory for building Gnuastro from the standard tarball release. If they are not present you cannot pass Gnuastro’s configuration step. The mandatory dependencies are therefore very basic (low-level) tools which are easy to obtain, build and install, see Section 3.1.1 [Mandatory dependencies], page 120, for a full discussion.

If you have the packages of Section 3.1.2 [Optional dependencies], page 122, Gnuastro will have additional functionality (for example converting FITS images to JPEG or PDF). If you are installing from a tarball as explained in Section 1.1 [Quick start], page 1, you can stop reading after this section. If you are cloning the version controlled source (see Section 3.2.2 [Version controlled source], page 132), an additional bootstrapping step is required before configuration and its dependencies are explained in Section 3.1.3 [Bootstrapping dependencies], page 124.

\(^1\) Section 3.1.3 [Bootstrapping dependencies], page 124, are required if you clone the full history.
Your operating system’s package manager is an easy and convenient way to download and install the dependencies that are already pre-built for your operating system. In Section 3.1.4 [Dependencies from package managers], page 128, we’ll list some common operating system package manager commands to install the optional and mandatory dependencies.

### 3.1.1 Mandatory dependencies

The mandatory Gnuastro dependencies are very basic and low-level tools. They all follow the same basic GNU based build system (like that shown in Section 1.1 [Quick start], page 1), so even if you don’t have them, installing them should be pretty straightforward. In this section we explain each program and any specific note that might be necessary in the installation.

#### 3.1.1.1 GNU Scientific Library

The GNU Scientific Library (http://www.gnu.org/software/gsl/), or GSL, is a large collection of functions that are very useful in scientific applications, for example integration, random number generation, and Fast Fourier Transform among many others. To install GSL from source, you can run the following commands after you have downloaded gsl-latest.tar.gz (http://ftpmirror.gnu.org/gsl/gsl-latest.tar.gz):

\[
\begin{align*}
\$ & \text{tar xf gsl-latest.tar.gz} \\
\$ & \text{cd gsl-X.X} \quad \# \text{Replace X.X with version number.} \\
\$ & \text{./configure} \\
\$ & \text{make -j8} \quad \# \text{Replace 8 with no. CPU threads.} \\
\$ & \text{make check} \\
\$ & \text{sudo make install}
\end{align*}
\]

#### 3.1.1.2 CFITSIO

CFITSIO (http://heasarc.gsfc.nasa.gov/fitsio/) is the closest you can get to the pixels in a FITS image while remaining faithful to the FITS standard (http://fits.gsfc.nasa.gov/fits_standard.html). It is written by William Pence, the principal author of the FITS standard\(^2\), and is regularly updated. Setting the definitions for all other software packages using FITS images.

Some GNU/Linux distributions have CFITSIO in their package managers, if it is available and updated, you can use it. One problem that might occur is that CFITSIO might not be configured with the `--enable-reentrant` option by the distribution. This option allows CFITSIO to open a file in multiple threads, it can thus provide great speed improvements. If CFITSIO was not configured with this option, any program which needs this capability will warn you and abort when you ask for multiple threads (see Section 4.4 [Multi-threaded operations], page 175).

To install CFITSIO from source, we strongly recommend that you have a look through Chapter 2 (Creating the CFITSIO library) of the CFITSIO manual and understand the options you can pass to `./configure` (they aren’t too much). This is a very basic package for most astronomical software and it is best that you configure it nicely with your system.

Once you download the source and unpack it, the following configure script should be enough for most purposes. Don’t forget to read chapter two of the manual though, for example the second option is only for 64bit systems. The manual also explains how to check if it has been installed correctly.

CFITSIO comes with two executable files called `fpack` and `funpack`. From their manual: they “are standalone programs for compressing and uncompressing images and tables that are stored in the FITS (Flexible Image Transport System) data format. They are analogous to the gzip and gunzip compression programs except that they are optimized for the types of astronomical images that are often stored in FITS format”. The commands below will compile and install them on your system along with CFITSIO. They are not essential for Gnuastro, since they are just wrappers for functions within CFITSIO, but they can come in handy. The `make utils` command is only available for versions above 3.39, it will build these executable files along with several other executable test files which are deleted in the following commands before the installation (otherwise the test files will also be installed).

The commands necessary to decompress, build and install CFITSIO from source are described below. Let’s assume you have downloaded `cfitsio_latest.tar.gz` (http://heasarc.gsfc.nasa.gov/FTP/software/fitsio/c/cfitsio_latest.tar.gz) and are in the same directory:

```
$ tar xf cfitsio_latest.tar.gz
$ cd cfitsio-X.XX # Replace X.XX with version
$ ./configure --prefix=/usr/local --enable-sse2 --enable-reentrant
$ make
$ make utils
$ ./testprog > testprog.lis # See below if this has an error
$ diff testprog.lis testprog.out # Should have no output
$ cmp testprog.fit testprog.std # Should have no output
$ rm cookbook fitscopy imcopy smem speed testprog
$ sudo make install
```

In the `./testprog > testprog.lis` step, you may confront an error, complaining that it can’t find `libcfitsio.so.AAA` (where AAA is an integer). This is the library that you just built and haven’t yet installed. But unfortunately some versions of CFITSIO don’t account for this on some OSs. To fix the problem, you need to tell your OS to also look into current CFITSIO build directory with the first command below, afterwards, the problematic command (second below) should run properly.

```
$ export LD_LIBRARY_PATH="$(pwd):$LD_LIBRARY_PATH"
$ ./testprog > testprog.lis
```

Recall that the modification above is ONLY NECESSARY FOR THIS STEP. Don’t put the `LD_LIBRARY_PATH` modification command in a permanent place (like your bash startup file). After installing CFITSIO, close your terminal and continue working on a new terminal (so `LD_LIBRARY_PATH` has its default value). For more on `LD_LIBRARY_PATH`, see Section 3.3.1.2 [Installation directory], page 139.

### 3.1.1.3 WCSLIB

WCSLIB (http://www.atnf.csiro.au/people/mcalabre/WCS/) is written and maintained by one of the authors of the World Coordinate System (WCS) definition in the
Chapter 3: Installation

FITS standard (http://fits.gsfc.nasa.gov/fits_standard.html), Mark Calabretta. It might be already built and ready in your distribution’s package management system. However, here the installation from source is explained, for the advantages of installation from source please see Section 3.1.1 [Mandatory dependencies], page 120. To install WCSLIB you will need to have CFITSIO already installed, see Section 3.1.1.2 [CFITSIO], page 120.

WCSLIB also has plotting capabilities which use PGPLOT (a plotting library for C). If you want to use those capabilities in WCSLIB, Section B.3 [PGPLOT], page 703, provides the PGPLOT installation instructions. However PGPLOT is old, so its installation is not easy, there are also many great modern WCS plotting tools (mostly in written in Python). Hence, if you will not be using those plotting functions in WCSLIB, you can configure it with the --without-pgplot option as shown below.

If you have the cURL library on your system and you installed CFITSIO version 3.42 or later, you will need to also link with the cURL library at configure time (through the -lcurl option as shown below). CFITSIO uses the cURL library for its HTTPS (or HTTP Secure) support and if it is present on your system, CFITSIO will depend on it. Therefore, if ./configure command below fails (you don’t have the cURL library), then remove this option and rerun it.

Let’s assume you have downloaded wcslib.tar.bz2 (ftp://ftp.atnf.csiro.au/pub/software/wcslib/wcslib.tar.bz2) and are in the same directory, to configure, build, check and install WCSLIB follow the steps below.

```
$ tar xf wcslib.tar.bz2

## In the 'cd' command, replace 'X.X' with version number.
$ cd wcslib-X.X

## If './configure' fails, remove '-lcurl' and run again.
$ ./configure LIBS="-pthread -lcurl -lm" --without-pgplot --disable-fortran

$ make
$ make check
$ sudo make install
```

### 3.1.2 Optional dependencies

The libraries listed here are only used for very specific applications, therefore they are optional and Gnuastro can be built without them (with only those specific features disabled). Since these are pretty low-level tools, they are not too hard to install from source, but you can also use your operating system’s package manager to easily install all of them. For more, see Section 3.1.4 [Dependencies from package managers], page 128.

If the ./configure script can’t find any of these optional dependencies, it will notify you of the operation(s) you can’t do due to not having them. If you continue the build and

---


4 As of early June 2016, its most recent version was uploaded in February 2001.

5 https://curl.haxx.se

6 https://en.wikipedia.org/wiki/HTTPS
request an operation that uses a missing library, Gnuastro’s programs will warn that the
optional library was missing at build-time and abort. Since Gnuastro was built without that
library, installing the library afterwards won’t help. The only way is to re-build Gnuastro
from scratch (after the library has been installed). However, for program dependencies (like
cURL or GhostScript) things are easier: you can install them after building Gnuastro also.
This is because libraries are used to build the internal structure of Gnuastro’s executables.
However, a program dependency is called by Gnuastro’s programs at run-time and has no
effect on their internal structure. So if a dependency program becomes available later, it
will be used next time it is requested.

GNU Libtool
Libtool is a program to simplify managing of the libraries to build an executable
(a program). GNU Libtool has some added functionality compared to other im-
plementations. If GNU Libtool isn’t present on your system at configuration
time, a warning will be printed and Section 11.2 [BuildProgram], page 521,
won’t be built or installed. The configure script will look into your search path
(PATH) for GNU Libtool through the following executable names: libtool (ac-
ceptable only if it is the GNU implementation) or glibtool. See Section 3.3.1.2
[Installation directory], page 139, for more on PATH.

GNU Libtool (the binary/executable file) is a low-level program that is prob-
ably already present on your system, and if not, is available in your operating
system package manager\(^7\). If you want to install GNU Libtool’s latest ver-
sion from source, please visit its web page (https://www.gnu.org/software/
libtool/).

Gnuastro’s tarball is shipped with an internal implementation of GNU Libtool.
Even if you have GNU Libtool, Gnuastro’s internal implementation is used for
the building and installation of Gnuastro. As a result, you can still build, install
and use Gnuastro even if you don’t have GNU Libtool installed on your system.
However, this internal Libtool does not get installed. Therefore, after Gnuas-
tro’s installation, if you want to use Section 11.2 [BuildProgram], page 521, to
compile and link your own C source code which uses the Section 11.3 [Gnuastro
library], page 525, you need to have GNU Libtool available on your system
(independent of Gnuastro). See Section 11.1 [Review of library fundamentals],
page 513, to learn more about libraries.

libgit2 Git is one of the most common version control systems (see Section 3.2.2 [Ver-
sion controlled source], page 132). When libgit2 is present, and Gnuastro’s
programs are run within a version controlled directory, outputs will contain the
version number of the working directory’s repository for future reproducibility.
See the COMMIT keyword header in Section 4.10 [Output FITS files], page 191,
for a discussion.

libjpeg libjpeg is only used by ConvertType to read from and write to JPEG images,
see Section 5.2.1 [Recognized file formats], page 211. libjpeg (http://www.injg.

\(^7\) Note that we want the binary/executable Libtool program which can be run on the command-line. In
Debian-based operating systems which separate various parts of a package, you want want libtool-bin,
the libtool package won’t contain the executable program.
org/) is a very basic library that provides tools to read and write JPEG images, most Unix-like graphic programs and libraries use it. Therefore you most probably already have it installed. libjpeg-turbo (http://libjpeg-turbo.virtualgl.org/) is an alternative to libjpeg. It uses Single instruction, multiple data (SIMD) instructions for ARM based systems that significantly decreases the processing time of JPEG compression and decompression algorithms.

libtiff libtiff is used by ConvertType and the libraries to read TIFF images, see Section 5.2.1 [Recognized file formats], page 211. libtiff (http://www.simplesystems.org/libtiff/) is a very basic library that provides tools to read and write TIFF images, most Unix-like operating system graphic programs and libraries use it. Therefore even if you don’t have it installed, it must be easily available in your package manager.

cURL cURL’s executable (curl) is called by Section 5.4 [Query], page 249, for submitting queries to remote datasets and retrieving the results. It isn’t necessary for the build of Gnuastro from source (only a warning will be printed if it can’t be found at configure time), so if you don’t have it at build-time there is no problem. Just be sure to have it when you run astquery, otherwise you’ll get an error about not finding curl.

GPL Ghostscript
GPL Ghostscript’s executable (gs) is called by ConvertType to compile a PDF file from a source PostScript file, see Section 5.2 [ConvertType], page 211. Therefore its headers (and libraries) are not needed.

SAO DS9 SAO DS9 (ds9) is a visualization tool for FITS images. Gnuastro’s astscript-fits-view program calls DS9 to visualize FITS images. We have a full appendix on it and how to install it in Section B.1 [SAO DS9], page 702. Since it is a run-time dependency, it can be installed at any later time (after building and installing Gnuastro).

TOPCAT TOPCAT (topcat) is a visualization tool for astronomical tables (most commonly: plotting). Gnuastro’s astscript-fits-view program calls TOPCAT it to visualize tables. We have a full appendix on it and how to install it in Section B.2 [TOPCAT], page 703. Since it is a run-time dependency, it can be installed at any later time (after building and installing Gnuastro).

3.1.3 Bootstrapping dependencies
Bootstrapping is only necessary if you have decided to obtain the full version controlled history of Gnuastro, see Section 3.2.2 [Version controlled source], page 132, and Section 3.2.2.1 [Bootstrapping], page 133. Using the version controlled source enables you to always be up to date with the most recent development work of Gnuastro (bug fixes, new functionalities, improved algorithms, etc). If you have downloaded a tarball (see Section 3.2 [Downloading the source], page 131), then you can ignore this subsection.

To successfully run the bootstrapping process, there are some additional dependencies to those discussed in the previous subsections. These are low level tools that are used by a large collection of Unix-like operating systems programs, therefore they are most probably
already available in your system. If they are not already installed, you should be able to easily find them in any GNU/Linux distribution package management system (\texttt{apt-get}, \texttt{yum}, \texttt{pacman}, etc). The short names in parenthesis in typewriter font after the package name can be used to search for them in your package manager. For the GNU Portability Library, GNU Autoconf Archive and \TeX Live, it is recommended to use the instructions here, not your operating system’s package manager.

GNU Portability Library (Gnulib)

To ensure portability for a wider range of operating systems (those that don’t include GNU C library, namely glibc), Gnuastro depends on the GNU portability library, or Gnulib. Gnulib keeps a copy of all the functions in glibc, implemented (as much as possible) to be portable to other operating systems. The bootstrap script can automatically clone Gnulib (as a gnulib/ directory inside Gnuastro), however, as described in Section 3.2.2.1 [Bootstrapping], page 133, this is not recommended.

The recommended way to bootstrap Gnuastro is to first clone Gnulib and the Autoconf archives (see below) into a local directory outside of Gnuastro. Let’s call it \texttt{DEVDIR} (which you can set to any directory). Currently in Gnuastro, both Gnulib and Autoconf archives have to be cloned in the same top directory:

\begin{verbatim}
$ DEVDIR=/home/yourname/Development
$ cd $DEVDIR
$ git clone https://git.sv.gnu.org/git/gnulib.git
$ git clone https://git.sv.gnu.org/git/autoconf-archive.git
\end{verbatim}

Gnulib is a source-based dependency of Gnuastro’s bootstrapping process, so simply having it is enough on your computer, there is no need to install, and thus check anything.

You now have the full version controlled source of these two repositories in separate directories. Both these packages are regularly updated, so every once in a while, you can run $ \texttt{git pull} within them to get any possible updates.

GNU Automake (\texttt{automake})

GNU Automake will build the \texttt{Makefile.in} files in each sub-directory using the (hand-written) \texttt{Makefile.am} files. The \texttt{Makefile.ins} are subsequently used to generate the \texttt{Makefiles} when the user runs \texttt{./configure} before building.

To check that you have a working GNU Automake in your system, you can try this command:

\begin{verbatim}
8 If you are not a developer in Gnulib or Autoconf archives, \texttt{DEVDIR} can be a directory that you don’t backup. In this way the large number of files in these projects won’t slow down your backup process or take bandwidth (if you backup to a remote server).
9 If you already have the Autoconf archives in a separate directory, or can’t clone it in the same directory as Gnulib, or you have it with another directory name (not \texttt{autoconf-archive/}), you can follow this short step. Set \texttt{AUTOCONFARCHIVES} to your desired address. Then define a symbolic link in \texttt{DEVDIR} with the following command so Gnuastro’s bootstrap script can find it:
10 If your internet connection is active, but Git complains about the network, it might be due to your network setup not recognizing the git protocol. In that case use the following URL for the HTTP protocol instead (for Autoconf archives, replace the name): http://git.sv.gnu.org/r/gnulib.git

\end{verbatim}
$ automake --version

GNU Autoconf (autoconf)

GNU Autoconf will build the configure script using the configurations we have defined (hand-written) in configure.ac.

To check that you have a working GNU Autoconf in your system, you can try this command:

$ autoconf --version

GNU Autoconf Archive

These are a large collection of tests that can be called to run at ./configure time. See the explanation under GNU Portability Library (Gnulib) above for instructions on obtaining it and keeping it up to date.

GNU Autoconf Archive is a source-based dependency of Gnuastro’s bootstrapping process, so simply having it is enough on your computer, there is no need to install, and thus check anything. Just don’t forget that it has to be in the same directory as Gnulib (described above).

GNU Texinfo (texinfo)

GNU Texinfo is the tool that formats this manual into the various output formats. To bootstrap Gnuastro you need all of Texinfo’s command-line programs. However, some operating systems package them separately, for example in Fedora, makeinfo is packaged in the texinfo-tex package.

To check that you have a working GNU Texinfo in your system, you can try this command:

$ makeinfo --version

GNU Libtool (libtool)

GNU Libtool is in charge of building all the libraries in Gnuastro. The libraries contain functions that are used by more than one program and are installed for use in other programs. They are thus put in a separate directory (lib/).

To check that you have a working GNU Libtool in your system, you can try this command (and from the output, make sure it is GNU’s libtool)

$ libtool --version

GNU help2man (help2man)

GNU help2man is used to convert the output of the --help option (Section 4.3.2 [--help], page 173) to the traditional Man page (Section 4.3.3 [Man pages], page 174).

To check that you have a working GNU Help2man in your system, you can try this command:

$ help2man --version

\LaTeX{} and some \TeX{} packages

Some of the figures in this book are built by \LaTeX{} (using the PGF/TikZ package). The \LaTeX{} source for those figures is version controlled for easy maintenance not the actual figures. So the ./bootstrap script will run \LaTeX{} to build the figures. The best way to install \LaTeX{} and all the necessary packages is
through \TeX\ Live (https://www.tug.org/texlive/) which is a package manager for \TeX\ related tools that is independent of any operating system. It is thus preferred to the \TeX\ Live versions distributed by your operating system. To install \TeX\ Live, go to the web page and download the appropriate installer by following the “download” link. Note that by default the full package repository will be downloaded and installed (around 4 Giga Bytes) which can take very long to download and to update later. However, most packages are not needed by everyone, it is easier, faster and better to install only the “Basic scheme” (consisting of only the most basic \TeX\ and \LaTeX\ packages, which is less than 200 Mega bytes).\footnote{You can also download the DVD iso file at a later time to keep as a backup for when you don’t have internet connection if you need a package.}

After the installation, be sure to set the environment variables as suggested in the end of the outputs. Any time you confront (need) a package you don’t have, simply install it with a command like below (similar to how you install software from your operating system’s package manager).\footnote{After running \TeX, or \LaTeX, you might get a warning complaining about a missingfile. Run ‘\texttt{tlmgr info missingfile}’ to see the package(s) containing that file which you can install.} To install all the necessary \TeX\ packages for a successful Gnuastro bootstrap, run this command:

```
$ su
# tlmgr install epsf jknapltx caption biblatex biber iftex \ etoolbox logreq xstring xkeyval pgf ms \ xcolor pgfplots times rsfs ps2eps epspdf
```

To check that you have a working \LaTeX\ executable in your system, you can try this command (this just checks if \LaTeX\ exists, as described above, if you have a missing package, you can easily identify it from the output and install it with \texttt{tlmgr}):

```
$ latex --version
```

ImageMagick (imagemagick)

ImageMagick is a wonderful and robust program for image manipulation on the command-line. bootstrap uses it to convert the book images into the formats necessary for the various book formats.

Since ImageMagick version 7, it is necessary to edit the policy file (/etc/ImageMagick-7/policy.xml) to have the following line (it maybe present, but commented, in this case un-comment it):

```xml
<policy domain="coder" rights="read|write" pattern="{PS,PDF,XPS}" />
```

If the following line is present, it is also necessary to comment/remove it.

```xml
<policy domain="delegate" rights="none" pattern="gs" />
```

To learn more about the ImageMagick security policy please see: https://imagemagick.org/script/security-policy.php.

To check that you have a working ImageMagick in your system, you can try this command:

```
$ convert --version
```
3.1.4 Dependencies from package managers

The most basic way to install a package on your system is to build the packages from source yourself. Alternatively, you can use your operating system’s package manager to download pre-compiled files and install them. The latter choice is easier and faster. However, we recommend that you build the Section 3.1.1 [Mandatory dependencies], page 120, yourself from source (all necessary commands and links are given in the respective section). Here are some basic reasons behind this recommendation.

1. Your operating system’s pre-built software might not be the most recent release. For example, Gnuastro itself is also packaged in some package managers. For the list see: https://repology.org/project/gnuastro/versions. You will notice that Gnuastro’s version in some operating systems is more than 10 versions old! It is the same for all the dependencies of Gnuastro.

2. For each package, Gnuastro might perform better (or require) certain configuration options that your distribution’s package managers didn’t add for you. If present, these configuration options are explained during the installation of each in the sections below (for example in Section 3.1.1.2 [CFITSIO], page 120). When the proper configuration has not been set, the programs should complain and inform you.

3. For the libraries, they might separate the binary file from the header files which can cause confusion, see Section 3.3.5 [Known issues], page 149.

4. Like any other tool, the science you derive from Gnuastro’s tools highly depend on these lower level dependencies, so generally it is much better to have a close connection with them. By reading their manuals, installing them and staying up to date with changes/bugs in them, your scientific results and understanding (of what is going on, and thus how you interpret your scientific results) will also correspondingly improve.

Based on your package manager, you can use any of the following commands to install the mandatory and optional dependencies. If your package manager isn’t included in the list below, please send us the respective command, so we add it.

As discussed above, we recommend installing the mandatory dependencies manually from source (see Section 3.1.1 [Mandatory dependencies], page 120). Therefore, in each command below, first the optional dependencies are given. The mandatory dependencies are included after an empty line. If you would also like to install the mandatory dependencies with your package manager, just ignore the empty line.

For better archivability and compression ratios, Gnuastro’s recommended tarball compression format is with the Lzip (http://lzip.nongnu.org/lzip.html) program, see Section 3.2.1 [Release tarball], page 131. Therefore, the package manager commands below also contain Lzip.

**apt-get** (Debian-based OSs: Debian, Ubuntu, Linux Mint, etc)

Debian (https://en.wikipedia.org/wiki/Debian) is one of the oldest GNU/Linux distributions\(^\text{13}\). It thus has a very extended user community and a robust internal structure and standards. All of it is free software and based on the work of volunteers around the world. Many distributions are thus derived from it, for example Ubuntu and Linux Mint. This arguably

\(^\text{13}\) https://en.wikipedia.org/wiki/List_of_Linux_distributions#Debian-based
makes Debian-based OSs the largest, and most used, class of GNU/Linux distributions. All of them use Debian’s Advanced Packaging Tool (APT, for example apt-get) for managing packages.

```
$ sudo apt-get install ghostscript libtool-bin libjpeg-dev \
   libtiff-dev libgit2-dev curl lzip \
   saods9 topcat \
   libgsl-dev libcfitsio-dev wcslib-dev
```

Gnuastro is packaged (https://tracker.debian.org/pkg/gnuastro) in Debian (and thus some of its derivate operating systems). Just make sure it is the most recent version.

dnf

yum (Red Hat-based OSs: Red Hat, Fedora, CentOS, Scientific Linux, etc)

Red Hat Enterprise Linux (https://en.wikipedia.org/wiki/Red_Hat) (RHEL) is released by Red Hat Inc. RHEL requires paid subscriptions for use of its binaries and support. But since it is free software, many other teams use its code to spin-off their own distributions based on RHEL. Red Hat-based GNU/Linux distributions initially used the “Yellowdog Updated, Modifier” (YUM) package manager, which has been replaced by “Dandified yum” (DNF). If the latter isn’t available on your system, you can use yum instead of dnf in the command below.

```
$ sudo dnf install ghostscript libtool libjpeg-devel \
   libtiff-devel libgit2-devel lzip curl \
   ds9 topcat \
   gsl-devel cfitsio-devel wcslib-devel
```

brew (macOS)

macOS (https://en.wikipedia.org/wiki/MacOS) is the operating system used on Apple devices. macOS does not come with a package manager pre-installed, but several widely used, third-party package managers exist, such as Homebrew or MacPorts. Both are free software. Currently we have only tested Gnuastro’s installation with Homebrew as described below.

If not already installed, first obtain Homebrew by following the instructions at https://brew.sh. Homebrew manages packages in different ‘taps’. To install WCSLIB (discussed in Section 3.1.1 [Mandatory dependencies], page 120) via Homebrew you will need to tap into brewsci/science first (the tap may change in the future, but can be found by calling brew search wcslib).

```
$ brew install ghostscript libtool libjpeg libtiff \
   libgit2 curl lzip saoimageds9 topcat \
   gsl cfitsio
$ brew tap brewsci/science
$ brew install wcslib
```
pacman (Arch Linux)

Arch Linux ([https://en.wikipedia.org/wiki/Arch_Linux](https://en.wikipedia.org/wiki/Arch_Linux)) is a smaller GNU/Linux distribution, which follows the KISS principle (“keep it simple, stupid”) as a general guideline. It “focuses on elegance, code correctness, minimalism and simplicity, and expects the user to be willing to make some effort to understand the system’s operation”. Arch GNU/Linux uses “Package manager” (Pacman) to manage its packages/components.

```
$ sudo pacman -S ghostscript libtool libjpeg libtiff \
   libgit2 curl lzip \
   gsl cfitsio wcslib
```

SAO DS9 and TOPCAT aren’t available in the standard Arch GNU/Linux repositories. However, installing and using both is very easy from their own web pages, as described in Section B.1 [SAO DS9], page 702, and Section B.2 [TOPCAT], page 703.

zypper (openSUSE and SUSE Linux Enterprise Server)

SUSE Linux Enterprise Server¹ (SLES) is the commercial offering which shares code and tools. Many additional packages are offered in the Build Service¹. openSUSE and SLES use zypper (cli) and YaST (GUI) for managing repositories and packages.

```
$ sudo zypper install ghostscript_any libtool pkgconfig \
   libcurl-devel libgit2-devel libjpeg62-devel \
   libtiff-devel curl ds9 topcat \
   gsl-devel cfitsio-devel wcslib-devel
```

When building Gnuastro, run the configure script with the following `CPPFLAGS` environment variable:

```
$ ./configure CPPFLAGS="-I/usr/include/cfitsio"
```

Usually, when libraries are installed by operating system package managers, there should be no problems when configuring and building other programs from source (that depend on the libraries: Gnuastro in this case). However, in some special conditions, problems may pop-up during the configuration, building, or checking/running any of Gnuastro’s programs. The most common of such problems and their solution are discussed below.

<table>
<thead>
<tr>
<th>Not finding library during configuration:</th>
<th>If a library is installed, but during Gnuastro’s configure step the library isn’t found, then configure Gnuastro like the command below (correcting <code>/path/to/lib</code>). For more, see Section 3.3.5 [Known issues], page 149, and Section 3.3.1.2 [Installation directory], page 139.</th>
</tr>
</thead>
<tbody>
<tr>
<td>$ .configure LDFLAGS=&quot;-L/path/to/lib&quot;</td>
<td></td>
</tr>
</tbody>
</table>

¹ [https://www.suse.com/products/server](https://www.suse.com/products/server)
² [https://build.opensuse.org](https://build.opensuse.org)
Chapter 3: Installation

Not finding header (.h) files while building: If a library is installed, but during Gnuastro’s `make` step, the library’s header (file with a `.h` suffix) isn’t found, then configure Gnuastro like the command below (correcting `/path/to/include`). For more, see Section 3.3.5 [Known issues], page 149, and Section 3.3.1.2 [Installation directory], page 139.

```bash
$ ./configure CPPFLAGS="-I/path/to/include"
```

Gnuastro’s programs don’t run during check or after install: If a library is installed, but the programs don’t run due to linking problems, set the `LD_LIBRARY_PATH` variable like below (assuming Gnuastro is installed in `/path/to/installed`). For more, see Section 3.3.5 [Known issues], page 149, and Section 3.3.1.2 [Installation directory], page 139.

```bash
$ export LD_LIBRARY_PATH="$LD_LIBRARY_PATH:/path/to/installed/lib"
```

3.2 Downloading the source

Gnuastro’s source code can be downloaded in two ways. As a tarball, ready to be configured and installed on your system (as described in Section 1.1 [Quick start], page 1), see Section 3.2.1 [Release tarball], page 131. If you want official releases of stable versions this is the best, easiest and most common option. Alternatively, you can clone the version controlled history of Gnuastro, run one extra bootstrapping step and then follow the same steps as the tarball. This will give you access to all the most recent work that will be included in the next release along with the full project history. The process is thoroughly introduced in Section 3.2.2 [Version controlled source], page 132.

3.2.1 Release tarball

A release tarball (commonly compressed) is the most common way of obtaining free and open source software. A tarball is a snapshot of one particular moment in the Gnuastro development history along with all the necessary files to configure, build, and install Gnuastro easily (see Section 1.1 [Quick start], page 1). It is very straightforward and needs the least set of dependencies (see Section 3.1.1 [Mandatory dependencies], page 120). Gnuastro has tarballs for official stable releases and pre-releases for testing. See Section 1.6 [Version numbering], page 7, for more on the two types of releases and the formats of the version numbers. The URLs for each type of release are given below.

This URL hosts the official stable releases of Gnuastro. Always use the most recent version (see Section 1.6 [Version numbering], page 7). By clicking on the “Last modified” title of the second column, the files will be sorted by their date which you can also use to find the latest version. It is recommended to use a mirror to download these tarballs, please visit http://ftpmirror.gnu.org/gnuastro/ and see below.

Pre-release tar-balls (http://alpha.gnu.org/gnu/gnuastro):
This URL contains unofficial pre-release versions of Gnuastro. The pre-release versions of Gnuastro here are for enthusiasts to try out before an official release. If there are problems, or bugs then the testers will inform the developers to fix before the next official release. See Section 1.6 [Version numbering], page 7, to understand how the version numbers here are formatted. If you want to re-
main even more up-to-date with the developing activities, please clone the version controlled source as described in Section 3.2.2 [Version controlled source], page 132.

Gnuastro’s official/stable tarball is released with two formats: Gzip (with suffix `.tar.gz`) and Lzip (with suffix `.tar.lz`). The pre-release tarballs (after version 0.3) are released only as an Lzip tarball. Gzip is a very well-known and widely used compression program created by GNU and available in most systems. However, Lzip provides a better compression ratio and more robust archival capacity. For example Gnuastro 0.3’s tarball was 2.9MB and 4.3MB with Lzip and Gzip respectively, see the Lzip web page (http://www.nongnu.org/lzip/lzip.html) for more. Lzip might not be pre-installed in your operating system, if so, installing it from your operating system’s package manager or from source is very easy and fast (it is a very small program).

The GNU FTP server is mirrored (has backups) in various locations on the globe (http://www.gnu.org/order/ftp.html). You can use the closest mirror to your location for a more faster download. Note that only some mirrors keep track of the pre-release (alpha) tarballs. Also note that if you want to download immediately after and announcement (see Section 1.10 [Announcements], page 14), the mirrors might need some time to synchronize with the main GNU FTP server.

### 3.2.2 Version controlled source

The publicly distributed Gnuastro tar-ball (for example `gnuastro-X.X.tar.gz`) does not contain the revision history, it is only a snapshot of the source code at one significant instant of Gnuastro’s history (specified by the version number, see Section 1.6 [Version numbering], page 7), ready to be configured and built. To be able to develop successfully, the revision history of the code can be very useful to track when something was added or changed, also some updates that are not yet officially released might be in it.

We use Git for the version control of Gnuastro. For those who are not familiar with it, we recommend the ProGit book (https://git-scm.com/book/en). The whole book is publicly available for online reading and downloading and does a wonderful job at explaining the concepts and best practices.

Let’s assume you want to keep Gnuastro in the `TOPGNUASTRO` directory (can be any directory, change the value below). The full version controlled history of Gnuastro can be cloned in `TOPGNUASTRO/-gnuastro` by running the following commands\(^\text{16}\):

\begin{verbatim}
$ TOPGNUASTRO=/home/yourname/Research/projects/
$ cd $TOPGNUASTRO
$ git clone git://git.sv.gnu.org/gnuastro.git
\end{verbatim}

The `$TOPGNUASTRO/-gnuastro` directory will contain hand-written (version controlled) source code for Gnuastro’s programs, libraries, this book and the tests. All are divided into sub-directories with standard and very descriptive names. The version controlled files in the top cloned directory are either mainly in capital letters (for example THANKS and README) or mainly written in small-caps (for example configure.ac and Makefile.am). The former are non-programming, standard writing for human readers containing

---

\(^{16}\) If your internet connection is active, but Git complains about the network, it might be due to your network setup not recognizing the Git protocol. In that case use the following URL which uses the HTTP protocol instead: http://git.sv.gnu.org/r/gnuastro.git
high-level information about the whole package. The latter are instructions to customize the GNU build system for Gnuastro. For more on Gnuastro’s source code structure, please see Chapter 12 [Developing], page 670. We won’t go any deeper here.

The cloned Gnuastro source cannot immediately be configured, compiled, or installed since it only contains hand-written files, not automatically generated or imported files which do all the hard work of the build process. See Section 3.2.2.1 [Bootstrapping], page 133, for the process of generating and importing those files (its not too hard!). Once you have bootstrapped Gnuastro, you can run the standard procedures (in Section 1.1 [Quick start], page 1). Very soon after you have cloned it, Gnuastro’s main master branch will be updated on the main repository (since the developers are actively working on Gnuastro), for the best practices in keeping your local history in sync with the main repository see Section 3.2.2.2 [Synchronizing], page 134.

### 3.2.2.1 Bootstrapping

The version controlled source code lacks the source files that we have not written or are automatically built. These automatically generated files are included in the distributed tar ball for each distribution (for example gnuastro-X.X.tar.gz, see Section 1.6 [Version numbering], page 7) and make it easy to immediately configure, build, and install Gnuastro. However from the perspective of version control, they are just bloatware and sources of confusion (since they are not changed by Gnuastro developers).

The process of automatically building and importing necessary files into the cloned directory is known as bootstrapping. After bootstrapping is done you are ready to follow the default GNU build steps that you normally run on the tarball (./configure && make for example, described more in Section 1.1 [Quick start], page 1). Some known issues with bootstrapping may occur during the process, to see how to fix them, please see Section 3.3.5 [Known issues], page 149.

All the instructions for an automatic bootstrapping are available in bootstrap and configured using bootstrap.conf. bootstrap and COPYING (which contains the software copyright notice) are the only files not written by Gnuastro developers but under version control to enable simple bootstrapping and legal information on usage immediately after cloning. bootstrap.conf is maintained by the GNU Portability Library (Gnulib) and this file is an identical copy, so do not make any changes in this file since it will be replaced when Gnulib releases an update. Make all your changes in bootstrap.conf.

The bootstrapping process has its own separate set of dependencies, the full list is given in Section 3.1.3 [Bootstrapping dependencies], page 124. They are generally very low-level and used by a very large set of commonly used programs, so they are probably already installed on your system. The simplest way to bootstrap Gnuastro is to simply run the bootstrap script within your cloned Gnuastro directory as shown below. However, please read the next paragraph before doing so (see Section 3.2.2 [Version controlled source], page 132, for TOPGNUASTRO).

```
$ cd TOPGNUASTRO/gnuastro
$ ./bootstrap # Requires internet connection
```

Without any options, bootstrap will clone Gnulib within your cloned Gnuastro directory (TOPGNUASTRO/gnuastro/gnulib) and download the necessary Autoconf archives macros. So if you run bootstrap like this, you will need an internet connection every time you decide
to bootstrap. Also, Gnuml is a large package and cloning it can be slow. It will also keep the full Gnuml repository within your Gnuastro repository, so if another one of your projects also needs Gnuml, and you insist on running bootstrap like this, you will have two copies. In case you regularly backup your important files, Gnuml will also slow down the backup process. Therefore while the simple invocation above can be used with no problem, it is not recommended. To do better, see the next paragraph.

The recommended way to get these two packages is thoroughly discussed in Section 3.1.3 [Bootstrapping dependencies], page 124, (in short: clone them in the separate NEVDIR/ directory). The following commands will take you into the cloned Gnuastro directory and run the bootstrap script, while telling it to copy some files (instead of making symbolic links, with the --copy option, this is not mandatory\(^\text{17}\)) and where to look for Gnuml (with the --gnuml-srcdir option). Please note that the address given to --gnuml-srcdir has to be an absolute address (so don’t use ~ or ../ for example).

\[
\begin{align*}
\text{$\$ cd $TOPGNUMASTRO/gnuastro} \\
\text{\$ ./bootstrap --copy --gnuml-srcdir=$DEVDIR/gnuml}
\end{align*}
\]

Since Gnuml and Autoconf archives are now available in your local directories, you don’t need an internet connection every time you decide to remove all untracked files and redo the bootstrap (see box below). You can also use the same command on any other project that uses Gnuml. All the necessary GNU C library functions, Autoconf macros and Automake inputs are now available along with the book figures. The standard GNU build system (Section 1.1 [Quick start], page 1) will do the rest of the job.

**Undoing the bootstrap:** During the development, it might happen that you want to remove all the automatically generated and imported files. In other words, you might want to reverse the bootstrap process. Fortunately Git has a good program for this job: `git clean`. Run the following command and every file that is not version controlled will be removed.

```
git clean -fxd
```

It is best to commit any recent change before running this command. You might have created new files since the last commit and if they haven’t been committed, they will all be gone forever (using `rm`). To get a list of the non-version controlled files instead of deleting them, add the n option to `git clean`, so it becomes `-fxdn`.

Besides the `bootstrap` and `bootstrap.conf`, the `bootstrapped/` directory and` README-hacking` file are also related to the bootstrapping process. The former hosts all the imported (bootstrapped) directories. Thus, in the version controlled source, it only contains a README file, but in the distributed tar-ball it also contains sub-directories filled with all bootstrapped files. `README-hacking` contains a summary of the bootstrapping process discussed in this section. It is a necessary reference when you haven’t built this book yet. It is thus not distributed in the Gnuastro tarball.

### 3.2.2.2 Synchronizing

The bootstrapping script (see Section 3.2.2.1 [Bootstrapping], page 133) is not regularly needed: you mainly need it after you have cloned Gnuastro (once) and whenever you want

\(^{17}\) The --copy option is recommended because some backup systems might do strange things with symbolic links.
to re-import the files from Gnulib, or Autoconf archives\(^{18}\) (not too common). However, Gnuastro developers are constantly working on Gnuastro and are pushing their changes to the official repository. Therefore, your local Gnuastro clone will soon be out-dated. Gnuastro has two mailing lists dedicated to its developing activities (see Section 12.11 [Developing mailing lists], page 692). Subscribing to them can help you decide when to synchronize with the official repository.

To pull all the most recent work in Gnuastro, run the following command from the top Gnuastro directory. If you don’t already have a built system, ignore `make distclean`. The separate steps are described in detail afterwards.

```
$ make distclean && git pull && autoreconf -f
```

You can also run the commands separately:

```
$ make distclean
$ git pull
$ autoreconf -f
```

If Gnuastro was already built in this directory, you don’t want some outputs from the previous version being mixed with outputs from the newly pulled work. Therefore, the first step is to clean/delete all the built files with `make distclean`. Fortunately the GNU build system allows the separation of source and built files (in separate directories). This is a great feature to keep your source directory clean and you can use it to avoid the cleaning step. Gnuastro comes with a script with some useful options for this job. It is useful if you regularly pull recent changes, see Section 3.3.2 [Separate build and source directories], page 145.

After the pull, we must re-configure Gnuastro with `autoreconf -f` (part of GNU Autoconf). It will update the `.configure` script and all the `Makefile.in`\(^{19}\) files based on the hand-written configurations (in `configure.ac` and the `Makefile.am` files). After running `autoreconf -f`, a warning about `TEXI2DVI` might show up, you can ignore that.

The most important reason for re-building Gnuastro’s build system is to generate/update the version number for your updated Gnuastro snapshot. This generated version number will include the commit information (see Section 1.6 [Version numbering], page 7). The version number is included in nearly all outputs of Gnuastro’s programs, therefore it is vital for reproducing an old result.

As a summary, be sure to run `autoreconf -f` after every change in the Git history. This includes synchronization with the main server or even a commit you have made yourself.

If you would like to see what has changed since you last synchronized your local clone, you can take the following steps instead of the simple command above (don’t type anything after `#`):

```
$ git checkout master       # Confirm if you are on master.
$ git fetch origin         # Fetch all new commits from server.
$ git log master..origin/master  # See all the new commit messages.
$ git merge origin/master  # Update your master branch.
```

---

\(^{18}\) https://savannah.gnu.org/task/index.php?13993 is defined for you to check if significant (for Gnuastro) updates are made in these repositories, since the last time you pulled from them.

\(^{19}\) In the GNU build system, `.configure` will use the `Makefile.in` files to create the necessary `Makefile` files that are later read by `make` to build the package.
$ autoreconf -f  # Update the build system.

By default `git log` prints the most recent commit first, add the `--reverse` option to see the changes chronologically. To see exactly what has been changed in the source code along with the commit message, add a `-p` option to the `git log`.

If you want to make changes in the code, have a look at Chapter 12 [Developing], page 670, to get started easily. Be sure to commit your changes in a separate branch (keep your `master` branch to follow the official repository) and re-run `autoreconf -f` after the commit. If you intend to send your work to us, you can safely use your commit since it will be ultimately recorded in Gnuastro’s official history. If not, please upload your separate branch to a public hosting service, for example Codeberg (https://codeberg.org), and link to it in your report/paper. Alternatively, run `make distcheck` and upload the output `gnuastro-X.X.X.XXX.tar.gz` to a publicly accessible web page so your results can be considered scientific (reproducible) later.

### 3.3 Build and install

This section is basically a longer explanation to the sequence of commands given in Section 1.1 [Quick start], page 1. If you didn’t have any problems during the Section 1.1 [Quick start], page 1, steps, you want to have all the programs of Gnuastro installed in your system, you don’t want to change the executable names during or after installation, you have root access to install the programs in the default system wide directory, the Letter paper size of the print book is fine for you or as a summary you don’t feel like going into the details when everything is working, you can safely skip this section.

If you have any of the above problems or you want to understand the details for a better control over your build and install, read along. The dependencies which you will need prior to configuring, building and installing Gnuastro are explained in Section 3.1 [Dependencies], page 119. The first three steps in Section 1.1 [Quick start], page 1, need no extra explanation, so we will skip them and start with an explanation of Gnuastro specific configuration options and a discussion on the installation directory in Section 3.3.1 [Configuring], page 136, followed by some smaller subsections: Section 3.3.3 [Tests], page 148, Section 3.3.4 [A4 print book], page 148, and Section 3.3.5 [Known issues], page 149, which explains the solutions to known problems you might encounter in the installation steps and ways you can solve them.

#### 3.3.1 Configuring

The `$ ./configure` step is the most important step in the build and install process. All the required packages, libraries, headers and environment variables are checked in this step. The behaviors of make and make install can also be set through command line options to this command.

The configure script accepts various arguments and options which enable the final user to highly customize whatever she is building. The options to configure are generally very similar to normal program options explained in Section 4.1.1 [Arguments and options], page 153. Similar to all GNU programs, you can get a full list of the options along with a short explanation by running

```
$ ./configure --help
```
A complete explanation is also included in the INSTALL file. Note that this file was written by the authors of GNU Autoconf (which builds the configure script), therefore it is common for all programs which use the $ ./configure script for building and installing, not just Gnuastro. Here we only discuss cases where you don’t have super-user access to the system and if you want to change the executable names. But before that, a review of the options to configure that are particular to Gnuastro are discussed.

### 3.3.1.1 Gnuastro configure options

Most of the options to configure (which are to do with building) are similar for every program which uses this script. Here the options that are particular to Gnuastro are discussed. The next topics explain the usage of other configure options which can be applied to any program using the GNU build system (through the configure script).

---

**--enable-debug**

Compile/build Gnuastro with debugging information, no optimization and without shared libraries.

In order to allow more efficient programs when using Gnuastro (after the installation), by default Gnuastro is built with a 3rd level (a very high level) optimization and no debugging information. By default, libraries are also built for static and shared linking (see Section 11.1.2 [Linking], page 517). However, when there are crashes or unexpected behavior, these three features can hinder the process of localizing the problem. This configuration option is identical to manually calling the configuration script with CFLAGS="-g -O0 --disable-shared".

In the (rare) situations where you need to do your debugging on the shared libraries, don’t use this option. Instead run the configure script by explicitly setting CFLAGS like this:

```
$ ./configure CFLAGS="-g -O0"
```

---

**--enable-check-with-valgrind**

Do the make check tests through Valgrind. Therefore, if any crashes or memory-related issues (segmentation faults in particular) occur in the tests, the output of Valgrind will also be put in the tests/test-suite.log file without having to manually modify the check scripts. This option will also activate Gnuastro’s debug mode (see the --enable-debug configure-time option described above).

Valgrind is free software. It is a program for easy checking of memory-related issues in programs. It runs a program within its own controlled environment and can thus identify the exact line-number in the program’s source where a memory-related issue occurs. However, it can significantly slow-down the tests. So this option is only useful when a segmentation fault is found during make check.

---

**--enable-progname**

Only build and install progname along with any other program that is enabled in this fashion. progname is the name of the executable without the ast, for example crop for Crop (with the executable name of astcrop).
Note that by default all the programs will be installed. This option (and the \texttt{--disable-progname} options) are only relevant when you don’t want to install all the programs. Therefore, if this option is called for any of the programs in Gnuastro, any program which is not explicitly enabled will not be built or installed.

\texttt{--disable-progname}\newline\texttt{--enable-progname=no}

Do not build or install the program named \texttt{progname}. This is very similar to the \texttt{--enable-progname}, but will build and install all the other programs except this one.

\begin{quote}
\textbf{Note:} If some programs are enabled and some are disabled, it is equivalent to simply enabling those that were enabled. Listing the disabled programs is redundant.
\end{quote}

\texttt{--enable-gnulibcheck}

Enable checks on the GNU Portability Library (Gnulib). Gnulib is used by Gnuastro to enable users of non-GNU based operating systems (that don’t use GNU C library or glibc) to compile and use the advanced features that this library provides. We make extensive use of such functions. If you give this option to \$ ./\texttt{configure}, when you run \$ \texttt{make check}, first the functions in Gnulib will be tested, then the Gnuastro executables. If your operating system does not support glibc or has an older version of it and you have problems in the build process (\$ \texttt{make}), you can give this flag to configure to see if the problem is caused by Gnulib not supporting your operating system or Gnuastro, see Section 3.3.5 [Known issues], page 149.

\texttt{--disable-guide-message}\newline\texttt{--enable-guide-message=no}

Do not print a guiding message during the GNU Build process of Section 1.1 [Quick start], page 1. By default, after each step, a message is printed guiding the user what the next command should be. Therefore, after ./\texttt{configure}, it will suggest running \texttt{make}. After \texttt{make}, it will suggest running \texttt{make check} and so on. If Gnuastro is configured with this option, for example

\$ ./\texttt{configure} \texttt{--disable-guide-message}

Then these messages will not be printed after any step (like most programs). For people who are not yet fully accustomed to this build system, these guidelines can be very useful and encouraging. However, if you find those messages annoying, use this option.

\texttt{--without-libgit2}

Build Gnuastro without libgit2 (for including Git commit hashes in output files), see Section 3.1.2 [Optional dependencies], page 122. libgit2 is an optional dependency, with this option, Gnuastro will ignore any possibly existing libgit2 that may already be on the system.
--without-libjpeg
Build Gnuastro without libjpeg (for reading/writing to JPEG files), see Section 3.1.2 [Optional dependencies], page 122. libjpeg is an optional dependency, with this option, Gnuastro will ignore any possibly existing libjpeg that may already be on the system.

--without-libtiff
Build Gnuastro without libtiff (for reading/writing to TIFF files), see Section 3.1.2 [Optional dependencies], page 122. libtiff is an optional dependency, with this option, Gnuastro will ignore any possibly existing libtiff that may already be on the system.

The tests of some programs might depend on the outputs of the tests of other programs. For example MakeProfiles is one the first programs to be tested when you run $ make check. MakeProfiles' test outputs (FITS images) are inputs to many other programs (which in turn provide inputs for other programs). Therefore, if you don’t install MakeProfiles for example, the tests for many the other programs will be skipped. To avoid this, in one run, you can install all the programs and run the tests but not install. If everything is working correctly, you can run configure again with only the programs you want. However, don’t run the tests and directly install after building.

3.3.1.2 Installation directory
One of the most commonly used options to ./configure is --prefix, it is used to define the directory that will host all the installed files (or the “prefix” in their final absolute file name). For example, when you are using a server and you don’t have administrator or root access. In this example scenario, if you don’t use the --prefix option, you won’t be able to install the built files and thus access them from anywhere without having to worry about where they are installed. However, once you prepare your startup file to look into the proper place (as discussed thoroughly below), you will be able to easily use this option and benefit from any software you want to install without having to ask the system administrators or install and use a different version of a software that is already installed on the server.

The most basic way to run an executable is to explicitly write its full file name (including all the directory information) and run it. One example is running the configuration script with the $ ./configure command (see Section 1.1 [Quick start], page 1). By giving a specific directory (the current directory or ./), we are explicitly telling the shell to look in the current directory for an executable file named ‘configure’. Directly specifying the directory is thus useful for executables in the current (or nearby) directories. However, when the program (an executable file) is to be used a lot, specifying all those directories will become a significant burden. For example, the ls executable lists the contents in a given directory and it is (usually) installed in the /usr/bin/ directory by the operating system maintainers. Therefore, if using the full address was the only way to access an executable, each time you wanted a listing of a directory, you would have to run the following command (which is very inconvenient, both in writing and in remembering the various directories).

$ /usr/bin/ls

To address this problem, we have the PATH environment variable. To understand it better, we will start with a short introduction to the shell variables. Shell variable values are basically treated as strings of characters. For example, it doesn’t matter if the value
is a name (string of *alphabetical* characters), or a number (string of *numeric* characters), or both. You can define a variable and a value for it by running

```bash
$ myvariable1=a_test_value
$ myvariable2="a test value"
```

As you see above, if the value contains white space characters, you have to put the whole value (including white space characters) in double quotes (`"`). You can see the value it represents by running

```bash
$ echo $myvariable1
$ echo $myvariable2
```

If a variable has no value or it wasn’t defined, the last command will only print an empty line. A variable defined like this will be known as long as this shell or terminal is running. Other terminals will have no idea it existed. The main advantage of shell variables is that if they are exported\(^\text{20}\), subsequent programs that are run within that shell can access their value. So by changing their value, you can change the “environment” of a program which uses them. The shell variables which are accessed by programs are therefore known as “environment variables”\(^\text{21}\). You can see the full list of exported variables that your shell recognizes by running:

```bash
$ printenv
```

HOME is one commonly used environment variable, it is any user’s (the one that is logged in) top directory. Try finding it in the command above. It is used so often that the shell has a special expansion (alternative) for it: `~`. Whenever you see file names starting with the tilde sign, it actually represents the value to the HOME environment variable, so `~/doc` is the same as `$HOME/doc`.

Another one of the most commonly used environment variables is PATH, it is a list of directories to search for executable names. Its value is a list of directories (separated by a colon, or `:`). When the address of the executable is not explicitly given (like `.configure` above), the system will look for the executable in the directories specified by PATH. If you have a computer nearby, try running the following command to see which directories your system will look into when it is searching for executable (binary) files, one example is printed here (notice how `/usr/bin`, in the `ls` example above, is one of the directories in PATH):

```bash
$ echo $PATH
/usr/local/sbin:/usr/local/bin:/usr/bin
```

By default PATH usually contains system-wide directories, which are readable (but not writable) by all users, like the above example. Therefore if you don’t have root (or administrator) access, you need to add another directory to PATH which you actually have write access to. The standard directory where you can keep installed files (not just executables) for your own user is the `~/.local/` directory. The names of hidden files start with a `.` (dot), so it will not show up in your common command-line listings, or on the graphical user interface. You can use any other directory, but this is the most recognized.

The top installation directory will be used to keep all the package’s components: programs (executables), libraries, include (header) files, shared data (like manuals), or config-

\(^{20}\) By running `$ export myvariable=a_test_value` instead of the simpler case in the text

\(^{21}\) You can use shell variables for other actions too, for example to temporarily keep some names or run loops on some files.
ration files (see Section 11.1 [Review of library fundamentals], page 513, for a thorough introduction to headers and linking). So it commonly has some of the following sub-directories for each class of installed components respectively: bin/, lib/, include/ man/, share/, etc/. Since the PATH variable is only used for executables, you can add the "/.local/bin" directory (which keeps the executables/programs or more generally, “binary” files) to PATH with the following command. As defined below, first the existing value of PATH is used, then your given directory is added to its end and the combined value is put back in PATH (run "$ echo $PATH" afterwards to check if it was added).

\[ \text{\$ PATH} = \text{\$PATH:} \\
\text{"/.local/bin} \]

Any executable that you installed in "/.local/bin" will now be usable without having to remember and write its full address. However, as soon as you leave/close your current terminal session, this modified PATH variable will be forgotten. Adding the directories which contain executables to the PATH environment variable each time you start a terminal is also very inconvenient and prone to errors. Fortunately, there are standard ‘startup files’ defined by your shell precisely for this (and other) purposes. There is a special startup file for every significant starting step:

/etc/profile and everything in /etc/profile.d/

These startup scripts are called when your whole system starts (for example after you turn on your computer). Therefore you need administrator or root privileges to access or modify them.

`~/.bash_profile`

If you are using (GNU) Bash as your shell, the commands in this file are run, when you log in to your account through Bash. Most commonly when you login through the virtual console (where there is no graphic user interface).

`~/.bashrc`

If you are using (GNU) Bash as your shell, the commands here will be run each time you start a terminal and are already logged in. For example, when you open your terminal emulator in the graphic user interface.

For security reasons, it is highly recommended to directly type in your HOME directory value by hand in startup files instead of using variables. So in the following, let’s assume your user name is ‘name’ (so "~" may be replaced with /home/name). To add "~/.local/bin" to your PATH automatically on any startup file, you have to “export” the new value of PATH in the startup file that is most relevant to you by adding this line:

\[ \text{export PATH} = \text{\$PATH:} / \text{home/name/.local/bin} \]

Now that you know your system will look into "/.local/bin" for executables, you can tell Gnuastro’s configure script to install everything in the top "/.local" directory using the --prefix option. When you subsequently run $ make install, all the install-able files will be put in their respective directory under "/.local/ (the executables in "/.local/bin, the compiled library files in "/.local/lib, the library header files in "/.local/include and so on, to learn more about these different files, please see [Review of library fundamentals], page 513). Note that tilde ("~") expansion will not happen if you put a ‘=’ between --prefix and "/.local"\(^{22}\), so we have avoided the = character here which is optional in GNU-style options, see [Options], page 154.

\(^{22}\) If you insist on using ‘=’, you can use --prefix=$HOME/.local.


```bash
$ ./configure --prefix ~/.local
```

You can install everything (including libraries like GSL, CFITSIO, or WCSLIB which are Gnuastro's mandatory dependencies, see Section 3.1.1 [Mandatory dependencies], page 120) locally by configuring them as above. However, recall that PATH is only for executable files, not libraries and that libraries can also depend on other libraries. For example WCSLIB depends on CFITSIO and Gnuastro needs both. Therefore, when you installed a library in a non-recognized directory, you have to guide the program that depends on them to look into the necessary library and header file directories. To do that, you have to define the LDFLAGS and CPPFLAGS environment variables respectively. This can be done while calling ./configure as shown below:

```bash
$ ./configure LDFLAGS=-L/home/name/.local/lib \
CPPFLAGS=-I/home/name/.local/include \
--prefix ~/.local
```

It can be annoying/buggy to do this when configuring every software that depends on such libraries. Hence, you can define these two variables in the most relevant startup file (discussed above). The convention on using these variables doesn’t include a colon to separate values (as PATH-like variables do). They use white space characters and each value is prefixed with a compiler option\(^{23}\). Note the -L and -I above (see Section 4.1.1.2 [Options], page 154), for -I see Section 11.1.1 [Headers], page 514, and for -L, see Section 11.1.2 [Linking], page 517. Therefore we have to keep the value in double quotation signs to keep the white space characters and adding the following two lines to the startup file of choice:

```bash
export LDFLAGS="$LDFLAGS -L/home/name/.local/lib"
export CPPFLAGS="$CPPFLAGS -I/home/name/.local/include"
```

Dynamic libraries are linked to the executable every time you run a program that depends on them (see Section 11.1.2 [Linking], page 517, to fully understand this important concept). Hence dynamic libraries also require a special path variable called LD_LIBRARY_PATH (same formatting as PATH). To use programs that depend on these libraries, you need to add `~/.local/lib` to your LD_LIBRARY_PATH environment variable by adding the following line to the relevant start-up file:

```bash
export LD_LIBRARY_PATH=$LD_LIBRARY_PATH:/home/name/.local/lib
```

If you also want to access the Info (see Section 4.3.4 [Info], page 174) and man pages (see Section 4.3.3 [Man pages], page 174) documentations add `~/.local/share/info` and `~/.local/share/man` to your INFOPATH\(^24\) and MANPATH environment variables respectively.

A final note is that order matters in the directories that are searched for all the variables discussed above. In the examples above, the new directory was added after the system specified directories. So if the program, library or manuals are found in the system wide directories, the user directory is no longer searched. If you want to search your local

---

\(^{23}\) These variables are ultimately used as options while building the programs. Therefore every value has be an option name followed be a value as discussed in Section 4.1.1.2 [Options], page 154.

\(^{24}\) Info has the following convention: “If the value of INFOPATH ends with a colon [or it isn't defined] ..., the initial list of directories is constructed by appending the build-time default to the value of INFOPATH.” So when installing in a non-standard directory and if INFOPATH was not initially defined, add a colon to the end of INFOPATH as shown below. Otherwise Info will not be able to find system-wide installed documentation:

```bash
echo 'export INFOPATH=$INFOPATH:/home/name/.local/share/info:' >> ~/.bashrc
```

Note that this is only an internal convention of Info: do not use it for other *PATH variables.
installation first, put the new directory before the already existing list, like the example below.

    export LD_LIBRARY_PATH=/home/name/.local/lib:$LD_LIBRARY_PATH

This is good when a library, for example CFITSIO, is already present on the system, but the system-wide install wasn’t configured with the correct configuration flags (see Section 3.1.1.2 [CFITSIO], page 120), or you want to use a newer version and you don’t have administrator or root access to update it on the whole system/server. If you update LD_LIBRARY_PATH by placing ~/.local/lib first (like above), the linker will first find the CFITSIO you installed for yourself and link with it. It thus will never reach the system-wide installation.

There are important security problems with using local installations first: all important system-wide executables and libraries (important executables like ls and cp, or libraries like the C library) can be replaced by non-secure versions with the same file names and put in the customized directory (~/.local in this example). So if you choose to search in your customized directory first, please be sure to keep it clean from executables or libraries with the same names as important system programs or libraries.

Summary: When you are using a server which doesn’t give you administrator/root access AND you would like to give priority to your own built programs and libraries, not the version that is (possibly already) present on the server, add these lines to your startup file. See above for which startup file is best for your case and for a detailed explanation on each. Don’t forget to replace ‘/YOUR-HOME-DIR’ with your home directory (for example ‘/home/your-id’):

    export PATH=’/YOUR-HOME-DIR/.local/bin:$PATH"
    export LDFLAGS="-L/YOUR-HOME-DIR/.local/lib $LDFLAGS"
    export MANPATH=’/YOUR-HOME-DIR/.local/share/man:$MANPATH"
    export CPPFLAGS="-I/YOUR-HOME-DIR/.local/include $CPPFLAGS"
    export INFOPATH=’/YOUR-HOME-DIR/.local/share/info:$INFOPATH"
    export LD_LIBRARY_PATH=’/YOUR-HOME-DIR/.local/lib:$LD_LIBRARY_PATH"

Afterwards, you just need to add an extra --prefix=/YOUR-HOME-DIR/.local to the ./configure command of the software that you intend to install. Everything else will be the same as a standard build and install, see Section 1.1 [Quick start], page 1.

3.3.1.3 Executable names

At first sight, the names of the executables for each program might seem to be uncommonly long, for example astnoisechisel or astcrop. We could have chosen terse (and cryptic) names like most programs do. We chose this complete naming convention (something like the commands in \TeX) so you don’t have to spend too much time remembering what the name of a specific program was. Such complete names also enable you to easily search for the programs.

To facilitate typing the names in, we suggest using the shell auto-complete. With this facility you can find the executable you want very easily. It is very similar to file name completion in the shell. For example, simply by typing the letters below (where [TAB] stands for the Tab key on your keyboard)

    $ ast [TAB] [TAB]
you will get the list of all the available executables that start with `ast` in your `PATH` environment variable directories. So, all the Gnuastro executables installed on your system will be listed. Typing the next letter for the specific program you want along with a Tab, will limit this list until you get to your desired program.

In case all of this does not convince you and you still want to type short names, some suggestions are given below. You should have in mind though, that if you are writing a shell script that you might want to pass on to others, it is best to use the standard name because other users might not have adopted the same customization. The long names also serve as a form of documentation in such scripts. A similar reasoning can be given for option names in scripts: it is good practice to always use the long formats of the options in shell scripts, see Section 4.1.1.2 [Options], page 154.

The simplest solution is making a symbolic link to the actual executable. For example let’s assume you want to type `ic` to run Crop instead of `astcrop`. Assuming you installed Gnuastro executables in `/usr/local/bin` (default) you can do this simply by running the following command as root:

```
# ln -s /usr/local/bin/astcrop /usr/local/bin/ic
```

In case you update Gnuastro and a new version of Crop is installed, the default executable name is the same, so your custom symbolic link still works.

The installed executable names can also be set using options to `$ ./configure`, see Section 3.3.1 [Configuring], page 136. GNU Autoconf (which configures Gnuastro for your particular system), allows the builder to change the name of programs with the three options `--program-prefix`, `--program-suffix` and `--program-transform-name`. The first two are for adding a fixed prefix or suffix to all the programs that will be installed. This will actually make all the names longer! You can use it to add versions of program names to the programs in order to simultaneously have two executable versions of a program.

The third configure option allows you to set the executable name at install time using the SED program. SED is a very useful ‘stream editor’. There are various resources on the internet to use it effectively. However, we should caution that using configure options will change the actual executable name of the installed program and on every re-install (an update for example), you have to also add this option to keep the old executable name updated. Also note that the documentation or configuration files do not change from their standard names either.

For example, let’s assume that typing `ast` on every invocation of every program is really annoying you! You can remove this prefix from all the executables at configure time by adding this option:

```
$ ./configure --program-transform-name='s/ast/ /
```

### 3.3.1.4 Configure and build in RAM

Gnuastro’s configure and build process (the GNU build system) involves the creation, reading, and modification of a large number of files (input/output, or I/O). Therefore file I/O issues can directly affect the work of developers who need to configure and build Gnuastro numerous times. Some of these issues are listed below:

- I/O will cause wear and tear on both the HDDs (mechanical failures) and SSDs (decreasing the lifetime).
Having the built files mixed with the source files can greatly affect backing up (syn-
chronization) of source files (since it involves the management of a large number of
small files that are regularly changed. Backup software can of course be configured to
ignore the built files and directories. However, since the built files are mixed with the
source files and can have a large variety, this will require a high level of customization.

One solution to address both these problems is to use the tmpfs file system (https://
en.wikipedia.org/wiki/Tmpfs). Any file in tmpfs is actually stored in the RAM (and
possibly SWAP), not on HDDs or SSDs. The RAM is built for extensive and fast I/O.
Therefore the large number of file I/Os associated with configuring and building will not
harm the HDDs or SSDs. Due to the volatile nature of RAM, files in the tmpfs file-system
will be permanently lost after a power-off. Since all configured and built files are derivative
files (not files that have been directly written by hand) there is no problem in this and this
feature can be considered as an automatic cleanup.

The modern GNU C library (and thus the Linux kernel) defines the /dev/shm directory
for this purpose in the RAM (POSIX shared memory). To build in it, you can use the
GNU build system’s ability to build in a separate directory (not necessarily in the source
directory) as shown below. Just set SRCDIR as the address of Gnuastro’s top source directory
(for example, the unpacked tarball).

$ mkdir /dev/shm/tmp-gnuastro-build
$ cd /dev/shm/tmp-gnuastro-build
$ SRCDIR/configure --srcdir=SRCDIR
$ make

Gnuastro comes with a script to simplify this process of configuring and building in a
different directory (a “clean” build), for more see Section 3.3.2 [Separate build and source
directories], page 145.

3.3.2 Separate build and source directories

The simple steps of Section 1.1 [Quick start], page 1, will mix the source and built files.
This can cause inconvenience for developers or enthusiasts following the most recent work
(see Section 3.2.2 [Version controlled source], page 132). The current section is mainly
focused on this later group of Gnuastro users. If you just install Gnuastro on major releases
(following Section 1.10 [Announcements], page 14), you can safely ignore this section.

When it is necessary to keep the source (which is under version control), but not the
derivative (built) files (after checking or installing), the best solution is to keep the source
and the built files in separate directories. One application of this is already discussed in
Section 3.3.1.4 [Configure and build in RAM], page 144.

To facilitate this process of configuring and building in a separate directory, Gnuastro
comes with the developer-build script. It is available in the top source directory and is
not installed. It will make a directory under a given top-level directory (given to --top-
build-dir) and build Gnuastro in there directory. It thus keeps the source completely
separated from the built files. For easy access to the built files, it also makes a symbolic
link to the built directory in the top source files called build.

When run without any options, default values will be used for its configuration. As with
Gnuastro’s programs, you can inspect the default values with -P (or --printparams, the
output just looks a little different here). The default top-level build directory is /dev/shm:
the shared memory directory in RAM on GNU/Linux systems as described in Section 3.3.1.4 [Configure and build in RAM], page 144.

Besides these, it also has some features to facilitate the job of developers or bleeding edge users like the \texttt{--debug} option to do a fast build, with debug information, no optimization, and no shared libraries. Here is the full list of options you can feed to this script to configure its operations.

\begin{quote}
\textbf{Not all Gnuastro’s common program behavior usable here: developer-build is just a non-installed script with a very limited scope as described above. It thus doesn’t have all the common option behaviors or configuration files for example.}
\end{quote}

\begin{quote}
\textbf{White space between option and value: developer-build doesn’t accept an = sign between the options and their values. It also needs at least one character between the option and its value. Therefore \texttt{-n 4} or \texttt{--numthreads 4} are acceptable, while \texttt{-n4}, \texttt{-n=4}, or \texttt{--numthreads=4} aren’t. Finally multiple short option names cannot be merged: for example you can say \texttt{-c -n 4}, but unlike Gnuastro’s programs, \texttt{-cn4} is not acceptable.}
\end{quote}

\begin{quote}
\textbf{Reusable for other packages:} This script can be used in any software which is configured and built using the GNU Build System. Just copy it in the top source directory of that software and run it from there.
\end{quote}

\begin{itemize}
\item[-b STR]
\item[--top-build-dir STR]
  The top build directory to make a directory for the build. If this option isn’t called, the top build directory is \texttt{/dev/shm} (only available in GNU/Linux operating systems, see Section 3.3.1.4 [Configure and build in RAM], page 144).
\item[-V]
\item[--version]
  Print the version string of Gnuastro that will be used in the build. This string will be appended to the directory name containing the built files.
\item[-a]
\item[--autoreconf]
  Run \texttt{autoreconf -f} before building the package. In Gnuastro, this is necessary when a new commit has been made to the project history. In Gnuastro’s build system, the Git description will be used as the version, see Section 1.6 [Version numbering], page 7, and Section 3.2.2.2 [Synchronizing], page 134.
\item[-c]
\item[--clean]
  Delete the contents of the build directory (clean it) before starting the configuration and building of this run.
\end{itemize}

This is useful when you have recently pulled changes from the main Git repository, or committed a change your self and ran \texttt{autoreconf -f}, see Section 3.2.2.2 [Synchronizing], page 134. After running GNU Autoconf, the version will be updated and you need to do a clean build.
-d
--debug
Build with debugging flags (for example to use in GNU Debugger, also known as GDB, or Valgrind), disable optimization and also the building of shared libraries. Similar to running the configure script of below:

```
$ ./configure --enable-debug
```

Besides all the debugging advantages of building with this option, it will also be significantly speed up the build (at the cost of slower built programs). So when you are testing something small or working on the build system itself, it will be much faster to test your work with this option.

-v
--valgrind
Build all make check tests within Valgrind. For more, see the description of --enable-check-with-valgrind in Section 3.3.1.1 [Gnuastro configure options], page 137.

-j INT
--jobs INT
The maximum number of threads/jobs for Make to build at any moment. As the name suggests (Make has an identical option), the number given to this option is directly passed on to any call of Make with its -j option.

-C
--check
After finishing the build, also run make check. By default, make check isn’t run because the developer usually has their own checks to work on (for example defined in tests/during-dev.sh).

-i
--install
After finishing the build, also run make install.

-D
--dist
Run make dist-lzip pdf to build a distribution tarball (in .tar.lz format) and a PDF manual. This can be useful for archiving, or sending to colleagues who don’t use Git for an easy build and manual.

-u STR
--upload STR
Activate the --dist (-D) option, then use secure copy (scp, part of the SSH tools) to copy the tarball and PDF to the src and pdf sub-directories of the specified server and its directory (value to this option). For example --upload my-server:dir, will copy the tarball in the dir/src, and the PDF manual in dir/pdf of my-server server. It will then make a symbolic link in the top server directory to the tarball that is called gnuastro-latest.tar.lz.

-P
--publish
Short for --autoreconf --clean --debug --check --upload STR. --debug is added because it will greatly speed up the build. It will have no effect on the produced tarball. This is good when you have made a commit and are ready to
publish it on your server (if nothing crashes). Recall that if any of the previous steps fail the script aborts.

-I
--install-archive
Short for --autoreconf --clean --check --install --dist. This is useful when you actually want to install the commit you just made (if the build and checks succeed). It will also produce a distribution tarball and PDF manual for easy access to the installed tarball on your system at a later time.

Ideally, Gnuastro’s Git version history makes it easy for a prepared system to revert back to a different point in history. But Gnuastro also needs to bootstrap files and also your collaborators might (usually do!) find it too much of a burden to do the bootstrapping themselves. So it is convenient to have a tarball and PDF manual of the version you have installed (and are using in your research) handily available.

-h
--help
-p
--printparams
Print a description of this script along with all the options and their current values.

3.3.3 Tests
After successfully building (compiling) the programs with the $ make command you can check the installation before installing. To run the tests, run

$ make check

For every program some tests are designed to check some possible operations. Running the command above will run those tests and give you a final report. If everything is OK and you have built all the programs, all the tests should pass. In case any of the tests fail, please have a look at Section 3.3.5 [Known issues], page 149, and if that still doesn’t fix your problem, look at the ./tests/test-suite.log file to see if the source of the error is something particular to your system or more general. If you feel it is general, please contact us because it might be a bug. Note that the tests of some programs depend on the outputs of other program’s tests, so if you have not installed them they might be skipped or fail. Prior to releasing every distribution all these tests are checked. If you have a reasonably modern terminal, the outputs of the successful tests will be colored green and the failed ones will be colored red.

These scripts can also act as a good set of examples for you to see how the programs are run. All the tests are in the tests/ directory. The tests for each program are shell scripts (ending with .sh) in a sub-directory of this directory with the same name as the program. See Section 12.7 [Test scripts], page 684, for more detailed information about these scripts in case you want to inspect them.

3.3.4 A4 print book
The default print version of this book is provided in the letter paper size. If you would like to have the print version of this book on paper and you are living in a country which uses
A4, then you can rebuild the book. The great thing about the GNU build system is that the book source code which is in Texinfo is also distributed with the program source code, enabling you to do such customization (hacking).

In order to change the paper size, you will need to have GNU Texinfo installed. Open doc/gnuastro.texi with any text editor. This is the source file that created this book. In the first few lines you will see this line:

@c@afourpaper

In Texinfo, a line is commented with @c. Therefore, un-comment this line by deleting the first two characters such that it changes to:

@afourpaper

Save the file and close it. You can now run the following command

$ make pdf

and the new PDF book will be available in SRCdir/doc/gnuastro.pdf. By changing the pdf in $ make pdf to ps or dvi you can have the book in those formats. Note that you can do this for any book that is in Texinfo format, they might not have @afourpaper line, so you can add it close to the top of the Texinfo source file.

### 3.3.5 Known issues

Depending on your operating system and the version of the compiler you are using, you might confront some known problems during the configuration ($ ./configure), compilation ($ make) and tests ($ make check). Here, their solutions are discussed.

- **$ ./configure**: Configure complains about not finding a library even though you have installed it. The possible solution is based on how you installed the package:
  - From your distribution’s package manager. Most probably this is because your distribution has separated the header files of a library from the library parts. Please also install the ‘development’ packages for those libraries too. Just add a `-dev` or `-devel` to the end of the package name and re-run the package manager. This will not happen if you install the libraries from source. When installed from source, the headers are also installed.
  - From source. Then your linker is not looking where you installed the library. If you followed the instructions in this chapter, all the libraries will be installed in `/usr/local/lib`. So you have to tell your linker to look in this directory. To do so, configure Gnuastro like this:

    $ ./configure LDFLAGS="-L/usr/local/lib"

    If you want to use the libraries for your other programming projects, then export this environment variable in a start-up script similar to the case for LD_LIBRARY_PATH explained below, also see Section 3.3.1.2 [Installation directory], page 139.

- **$ make**: Complains about an unknown function on a non-GNU based operating system.
  In this case, please run $ ./configure with the `--enable-gnulibcheck` option to see if the problem is from the GNU Portability Library (Gnulib) not supporting your system or if there is a problem in Gnuastro, see Section 3.3.1.1 [Gnuastro configure options], page 137. If the problem is not in Gnulib and after all its tests you get the same complaint from make, then please contact us at bug-gnuastro@gnu.org. The cause is probably that a function that we have used is not supported by your operating system.
and we didn’t included it along with the source tar ball. If the function is available in Gnumlib, it can be fixed immediately.

- **$ make**: *Can’t find the headers (.h files) of installed libraries.* Your C pre-processor (CPP) isn’t looking in the right place. To fix this, configure Gnuastro with an additional **CPPFLAGS** like below (assuming the library is installed in `/usr/local/include`):

  ```
  $ ./configure CPPFLAGS="-I/usr/local/include"
  ```

  If you want to use the libraries for your other programming projects, then export this environment variable in a start-up script similar to the case for **LD_LIBRARY_PATH** explained below, also see Section 3.3.1.2 [Installation directory], page 139.

- **$ make check**: *Only the first couple of tests pass, all the rest fail or get skipped.* It is highly likely that when searching for shared libraries, your system doesn’t look into the `/usr/local/lib` directory (or wherever you installed Gnuastro or its dependencies). To make sure it is added to the list of directories, add the following line to your `~/.bashrc` file and restart your terminal. Don’t forget to change `/usr/local/lib` if the libraries are installed in other (non-standard) directories.

  ```
  export LD_LIBRARY_PATH="$LD_LIBRARY_PATH:/usr/local/lib"
  ```

  You can also add more directories by using a colon `:` to separate them. See Section 3.3.1.2 [Installation directory], page 139, and Section 11.1.2 [Linking], page 517, to learn more on the **PATH** variables and dynamic linking respectively.

- **$ make check**: *The tests relying on external programs (for example fits2pdf.sh fail.)*

  This is probably due to the fact that the version number of the external programs is too old for the tests we have performed. Please update the program to a more recent version. For example to create a PDF image, you will need GPL Ghostscript, but older versions do not work, we have successfully tested it on version 9.15. Older versions might cause a failure in the test result.

- **$ make pdf**: *The PDF book cannot be made.* To make a PDF book, you need to have the GNU Texinfo program (like any program, the more recent the better). A working TeX program is also necessary, which you can get from Tex Live.

- After **make check**: do not copy the programs’ executables to another (for example, the installation) directory manually (using `cp`, or `mv` for example). In the default configuration, the program binaries need to link with Gnuastro’s shared library which is also built and installed with the programs. Therefore, to run successfully before and after installation, linking modifications need to be made by GNU Libtool at installation time. **make install** does this internally, but a simple copy might give linking errors when you run it. If you need to copy the executables, you can do so after installation.

- **$ make** (when bootstrapping): After you have bootstrapped Gnuastro from the version-controlled source, you may confront the following (or a similar) error when converting images (for more on bootstrapping, see Section 3.2.2.1 [Bootstrapping], page 133):

  ```
  convert: attempt to perform an operation not allowed by the security policy ‘gs’ error/delegate.c/ExternalDelegateCommand/378.
  ```

---

25 https://www.tug.org/texlive/

26 If you configure Gnuastro with the **--disable-shared** option, then the libraries will be statically linked to the programs and this problem won’t exist, see Section 11.1.2 [Linking], page 517.
This error is a known issue\textsuperscript{27} with \texttt{ImageMagick} security policies in some operating systems. In short, \texttt{imagemagick} uses Ghostscript for PDF, EPS, PS and XPS parsing. However, because some security vulnerabilities have been found in Ghostscript\textsuperscript{28}, by default, ImageMagick may be compiled without Ghostscript library. In such cases, if allowed, ImageMagick will fall back to the external \texttt{gs} command instead of the library. But this may be disabled with the following (or a similar) lines in \\
\texttt{/etc/ImageMagick-7/policy.xml} (anything related to PDF, PS, or GhostScript).

\begin{verbatim}
  <policy domain="delegate" rights="none" pattern="gs" />
  <policy domain="module" rights="none" pattern="\{PS,PDF,XPS\}" />
\end{verbatim}

To fix this problem, simply comment such lines (by placing a \texttt{<!--} before each statement/line and \texttt{-->} at the end of that statement/line).

If your problem was not listed above, please file a bug report (Section 1.8 [Report a bug], page 11).

\textsuperscript{27} \url{https://wiki.archlinux.org/title/ImageMagick}

\textsuperscript{28} \url{https://security.archlinux.org/package/ghostscript}
4 Common program behavior

All the programs in Gnuastro share a set of common behavior mainly to do with user interaction to facilitate their usage and development. This includes how to feed input datasets into the programs, how to configure them, specifying the outputs, numerical data types, treating columns of information in tables, etc. This chapter is devoted to describing this common behavior in all programs. Because the behaviors discussed here are common to several programs, they are not repeated in each program’s description.

In Section 4.1 [Command-line], page 152, a very general description of running the programs on the command-line is discussed, like difference between arguments and options, as well as options that are common/shared between all programs. None of Gnuastro’s programs keep any internal configuration value (values for their different operational steps), they read their configuration primarily from the command-line, then from specific files in directory, user, or system-wide settings. Using these configuration files can greatly help reproducible and robust usage of Gnuastro, see Section 4.2 [Configuration files], page 169, for more.

It is not possible to always have the different options and configurations of each program on the top of your head. It is very natural to forget the options of a program, their current default values, or how it should be run and what it did. Gnuastro’s programs have multiple ways to help you refresh your memory in multiple levels (just an option name, a short description, or fast access to the relevant section of the manual. See Section 4.3 [Getting help], page 172, for more on benefiting from this very convenient feature.

Many of the programs use the multi-threaded character of modern CPUs, in Section 4.4 [Multi-threaded operations], page 175, we’ll discuss how you can configure this behavior, along with some tips on making best use of them. In Section 4.5 [Numeric data types], page 178, we’ll review the various types to store numbers in your datasets: setting the proper type for the usage context\(^1\) can greatly improve the file size and also speed of reading, writing or processing them.

We’ll then look into the recognized table formats in Section 4.7 [Tables], page 183, and how large datasets are broken into tiles, or mesh grid in Section 4.8 [Tessellation], page 189. Finally, we’ll take a look at the behavior regarding output files: Section 4.9 [Automatic output], page 191, describes how the programs set a default name for their output when you don’t give one explicitly (using `--output`). When the output is a FITS file, all the programs also store some very useful information in the header that is discussed in Section 4.10 [Output FITS files], page 191.

4.1 Command-line

Gnuastro’s programs are customized through the standard Unix-like command-line environment and GNU style command-line options. Both are very common in many Unix-like operating system programs. In Section 4.1.1 [Arguments and options], page 153, we’ll start with the difference between arguments and options and elaborate on the GNU style of op-

\(^1\) For example if the values in your dataset can only be integers between 0 or 65000, store them in an unsigned 16-bit type, not 64-bit floating point type (which is the default in most systems). It takes four times less space and is much faster to process.
4.1.1 Arguments and options

When you type a command on the command-line, it is passed onto the shell (a generic name for the program that manages the command-line) as a string of characters. As an example, see the “Invoking ProgramName” sections in this manual for some examples of commands with each program, like Section 5.3.3 [Invoking Table], page 238, Section 5.1.1 [Invoking Fits], page 196, or Section 7.1.5 [Invoking Statistics], page 350.

The shell then breaks up your string into separate tokens or words using any metacharacters (like white-space, tab, 1, > or ;) that are in the string. On the command-line, the first thing you usually enter is the name of the program you want to run. After that, you can specify two types of tokens: arguments and options. In the GNU-style, arguments are those tokens that are not preceded by any hyphens (-, see Section 4.1.1.1 [Arguments], page 154). Here is one example:

```
$ astcrop --center=53.162551,-27.789676 -w10/3600 --mode=wcs udf.fits
```

In the example above, we are running Section 6.1 [Crop], page 259, to crop a region of width 10 arc-seconds centered at the given RA and Dec from the input Hubble Ultra-Deep Field (UDF) FITS image. Here, the argument is `udf.fits`. Arguments are most commonly the input file names containing your data. Options start with one or two hyphens, followed by an identifier for the option (the option’s name, for example, `--center`, `-w`, `--mode` in the example above) and its value (anything after the option name, or the optional = character). Through options you can configure how the program runs (interprets the data you provided).

Arguments can be mandatory and optional and unlike options, they don’t have any identifiers. Hence, when there multiple arguments, their order might also matter (for example in `cp` which is used for copying one file to another location). The outputs of `--usage` and `--help` shows which arguments are optional and which are mandatory, see Section 4.3.1 [--usage], page 172.

As their name suggests, options can be considered to be optional and most of the time, you don’t have to worry about what order you specify them in. When the order does matter, or the option can be invoked multiple times, it is explicitly mentioned in the “Invoking ProgramName” section of each program (this is a very important aspect of an option).

If there is only one such character, you can use a backslash (\) before it. If there are multiple, it might be easier to simply put your whole argument or option value inside of double quotes ("). In such cases, everything inside the double quotes will be seen as one token or word.

For example, let’s say you want to specify the header data unit (HDU) of your FITS file using a complex expression like ‘3; images(exposure > 100)’. If you simply add these after the `--hdu` (-h) option, the programs in Gnuastro will read the value to the HDU option as ‘3’ and run. Then, the shell will attempt to run a separate command ‘images(exposure > 100)’ and complain about a syntax error. This is because the semicolon (;) is an ‘end of command’ character in the shell. To solve this problem you can simply put double quotes around the whole string you want to pass to `--hdu` as seen below:

```
$ astcrop --hdu="3; images(exposure > 100)" image.fits
```
4.1.1.1 Arguments

In Gnuastro, arguments are almost exclusively used as the input data file names. Please consult the first few paragraphs of the “Invoking ProgramName” section for each program for a description of what it expects as input, how many arguments, or input data, it accepts, or in what order. Everything particular about how a program treats arguments, is explained under the “Invoking ProgramName” section for that program.

Generally, if there is a standard file name suffix for a particular format, that filename extension is checked to identify their format. In astronomy (and thus Gnuastro), FITS is the preferred format for inputs and outputs, so the focus here and throughout this book is on FITS. However, other formats are also accepted in special cases, for example Section 5.2 [ConvertType], page 211, also accepts JPEG or TIFF inputs, and writes JPEG, EPS or PDF files. The recognized suffixes for these formats are listed there.

The list below shows the recognized suffixes for FITS data files in Gnuastro’s programs. However, in some scenarios FITS writers may not append a suffix to the file, or use a non-recognized suffix (not in the list below). Therefore if a FITS file is expected, but it doesn’t have any of these suffixes, Gnuastro programs will look into the contents of the file and if it does conform with the FITS standard, the file will be used. Just note that checking about 5 characters at the end of a name string is much more efficient than opening and checking the contents of a file, so it is generally recommended to have a recognized FITS suffix.

- `.fits`: The standard file name ending of a FITS image.
- `.fit`: Alternative (3 character) FITS suffix.
- `.fits.gz`: A FITS image compressed with GNU zip (gzip).
- `.fits.fz`: A FITS image compressed with `fpack`.
- `.imh`: IRAF format image file.

Through out this book and in the command-line outputs, whenever we want to generalize all such astronomical data formats in a text place-holder, we will use `ASTRdata`, we will assume that the extension is also part of this name. Any file ending with these names is directly passed on to CFITSIO to read. Therefore you don’t necessarily have to have these files on your computer, they can also be located on an FTP or HTTP server too, see the CFITSIO manual for more information.

CFITSIO has its own error reporting techniques, if your input file(s) cannot be opened, or read, those errors will be printed prior to the final error by Gnuastro.

4.1.1.2 Options

Command-line options allow configuring the behavior of a program in all GNU/Linux applications for each particular execution on a particular input data. A single option can be called in two ways: `long` or `short`. All options in Gnuastro accept the long format which has two hyphens an can have many characters (for example `--hdus`). Short options only have one hyphen (`-`) followed by one character (for example `-h`). You can see some examples in the list of options in Section 4.1.2 [Common options], page 156, or those for each program’s “Invoking ProgramName” section. Both formats are shown for those which support both. First the short is shown then the long.
Usually, the short options are for when you are writing on the command-line and want to save keystrokes and time. The long options are good for shell scripts, where you aren’t usually rushing. Long options provide a level of documentation, since they are more descriptive and less cryptic. Usually after a few months of not running a program, the short options will be forgotten and reading your previously written script will not be easy.

Some options need to be given a value if they are called and some don’t. You can think of the latter type of options as on/off options. These two types of options can be distinguished using the output of the --help and --usage options, which are common to all GNU software, see Section 4.3 [Getting help], page 172. In Gnuastro we use the following strings to specify when the option needs a value and what format that value should be in. More specific tests will be done in the program and if the values are out of range (for example negative when the program only wants a positive value), an error will be reported.

**INT**  
The value is read as an integer.

**FLT**  
The value is read as a float. There are generally two types, depending on the context. If they are for fractions, they will have to be less than or equal to unity.

**STR**  
The value is read as a string of characters. For example column names in a table, or HDU names in a multi-extension FITS file. Other examples include human-readable settings by some programs like the --domain option of the Convolve program that can be either spatial or frequency (to specify the type of convolution, see Section 6.3 [Convolve], page 306).

**FITS or FITS/TXT**  
The value should be a file (most commonly FITS). In many cases, other formats may also be accepted (for example input tables can be FITS or plain-text, see Section 4.7.1 [Recognized table formats], page 184).

To specify a value in the short format, simply put the value after the option. Note that since the short options are only one character long, you don’t have to type anything between the option and its value. For example, for `-h2`, `-h 2`, `-hdu 2` or `-hdu=2` are all equivalent.

The short format of on/off options (those that don’t need values) can be concatenated for example these two hypothetical sequences of options are equivalent: `-a -b -c4` and `-abc4`. As an example, consider the following command to run Crop:

```
$ astcrop -Dr3 --width 3 catalog.txt --deccol=4 ASTRdata
```

The `$` is the shell prompt, `astcrop` is the program name. There are two arguments (catalog.txt and ASTRdata) and four options, two of them given in short format (`-D`, `-r`) and two in long format (`--width` and `--deccol`). Three of them require a value and one (`-D`) is an on/off option.

If an abbreviation is unique between all the options of a program, the long option names can be abbreviated. For example, instead of typing `--printparams`, typing `--print` or maybe even `--pri` will be enough, if there are conflicts, the program will warn you and show you the alternatives. Finally, if you want the argument parser to stop parsing arguments beyond a certain point, you can use two dashes: `--`. No text on the command-line beyond these two dashes will be parsed.
Gnuastro has two types of options with values, those that only take a single value are the most common type. If these options are repeated or called more than once on the command-line, the value of the last time it was called will be assigned to it. This is very useful when you are testing/experimenting. Let’s say you want to make a small modification to one option value. You can simply type the option with a new value in the end of the command and see how the script works. If you are satisfied with the change, you can remove the original option for human readability. If the change wasn’t satisfactory, you can remove the one you just added and not worry about forgetting the original value. Without this capability, you would have to memorize or save the original value somewhere else, run the command and then change the value again which is not at all convenient and is potentially cause lots of bugs.

On the other hand, some options can be called multiple times in one run of a program and can thus take multiple values (for example see the --column option in Section 5.3.3 [Invoking Table], page 238. In these cases, the order of stored values is the same order that you specified on the command-line.

Gnuastro’s programs don’t keep any internal default values, so some options are mandatory and if they don’t have a value, the program will complain and abort. Most programs have many such options and typing them by hand on every call is impractical. To facilitate the user experience, after parsing the command-line, Gnuastro’s programs read special configuration files to get the necessary values for the options you haven’t identified on the command-line. These configuration files are fully described in Section 4.2 [Configuration files], page 169.

CAUTION: In specifying a file address, if you want to use the shell’s tilde expansion (~) to specify your home directory, leave at least one space between the option name and your value. For example use -o ~/test, --output ~/test or --output=~/test. Calling them with -o~/test or --output=~/test will disable shell expansion.

CAUTION: If you forget to specify a value for an option which requires one, and that option is the last one, Gnuastro will warn you. But if it is in the middle of the command, it will take the text of the next option or argument as the value which can cause undefined behavior.

NOTE: In some contexts Gnuastro’s counting starts from 0 and in others 1. You can assume by default that counting starts from 1, if it starts from 0 for a special option, it will be explicitly mentioned.

4.1.2 Common options
To facilitate the job of the users and developers, all the programs in Gnuastro share some basic command-line options for the options that are common to many of the programs. The full list is classified as Section 4.1.2.1 [Input/Output options], page 157, Section 4.1.2.2 [Processing options], page 159, and Section 4.1.2.3 [Operating mode options], page 162. In some programs, some of the options are irrelevant, but still recognized (you won’t get an
unrecognized option error, but the value isn’t used). Unless otherwise mentioned, these options are identical between all programs.

4.1.2.1 Input/Output options

These options are to do with the input and outputs of the various programs.

--stdintimeout

Number of micro-seconds to wait for writing/typing in the first line of standard input from the command-line (see Section 4.1.4 [Standard input], page 167). This is only relevant for programs that also accept input from the standard input, and you want to manually write/type the contents on the terminal. When the standard input is already connected to a pipe (output of another program), there won’t be any waiting (hence no timeout, thus making this option redundant).

If the first line-break (for example with the ENTER key) is not provided before the timeout, the program will abort with an error that no input was given.

Note that this time interval is only for the first line that you type. Once the first line is given, the program will assume that more data will come and accept rest of your inputs without any time limit. You need to specify the ending of the standard input, for example by pressing CTRL-D after a new line.

Note that any input you write/type into a program on the command-line with Standard input will be discarded (lost) once the program is finished. It is only recoverable manually from your command-line (where you actually typed) as long as the terminal is open. So only use this feature when you are sure that you don’t need the dataset (or have a copy of it somewhere else).

-h STR/INT

--hdu=STR/INT

The name or number of the desired Header Data Unit, or HDU, in the FITS image. A FITS file can store multiple HDUs or extensions, each with either an image or a table or nothing at all (only a header). Note that counting of the extensions starts from 0(zero), not 1(one). Counting from 0 is forced on us by CFITSIO which directly reads the value you give with this option (see Section 3.1.1.2 [CFITSIO], page 120). When specifying the name, case is not important so IMAGE, image or ImAgE are equivalent.

CFITSIO has many capabilities to help you find the extension you want, far beyond the simple extension number and name. See CFITSIO manual’s “HDU Location Specification” section for a very complete explanation with several examples. A # is appended to the string you specify for the HDU2 and the result is put in square brackets and appended to the FITS file name before calling CFITSIO to read the contents of the HDU for all the programs in Gnuastro.

-s STR

--searchin=STR

Where to match/search for columns when the column identifier wasn’t a number, see Section 4.7.3 [Selecting table columns], page 188. The acceptable values

\[\text{With the } \# \text{ character, CFITSIO will only read the desired HDU into your memory, not all the existing HDUs in the fits file.}\]
are name, unit, or comment. This option is only relevant for programs that take table columns as input.

-I
--ignorecase
Ignore case while matching/searching column meta-data (in the field specified by the --searchin). The FITS standard suggests to treat the column names as case insensitive, which is strongly recommended here also but is not enforced. This option is only relevant for programs that take table columns as input.
This option is not relevant to Section 11.2 [BuildProgram], page 521, hence in that program the short option -I is used for include directories, not to ignore case.

-o STR
--output=STR
The name of the output file or directory. With this option the automatic output names explained in Section 4.9 [Automatic output], page 191, are ignored.

-T STR
--type=STR
The data type of the output depending on the program context. This option isn’t applicable to some programs like Section 5.1 [Fits], page 194, and will be ignored by them. The different acceptable values to this option are fully described in Section 4.5 [Numeric data types], page 178.

-D
--dontdelete
By default, if the output file already exists, Gnuastro’s programs will silently delete it and put their own outputs in its place. When this option is activated, if the output file already exists, the programs will not delete it, will warn you, and will abort.

-K
--keepinputdir
In automatic output names, don’t remove the directory information of the input file names. As explained in Section 4.9 [Automatic output], page 191, if no output name is specified (with --output), then the output name will be made in the existing directory based on your input’s file name (ignoring the directory of the input). If you call this option, the directory information of the input will be kept and the automatically generated output name will be in the same directory as the input (usually with a suffix added). Note that his is only relevant if you are running the program in a different directory than the input data.

-t STR
--tableformat=STR
The output table’s type. This option is only relevant when the output is a table and its format cannot be deduced from its filename. For example, if a name ending in .fits was given to --output, then the program knows you want a FITS table. But there are two types of FITS tables: FITS ASCII, and FITS
binaries. Thus, with this option, the program is able to identify which type you want. The currently recognized values to this option are:

```
-wcslinearmatrix=STR
```

Select the linear transformation matrix of the output’s WCS. This option only takes two values: `pc` (for the \( PC_{i,j} \) formalism) and `cd` (for \( CD_{i,j} \)). For more on the different formalisms, please see Section 8.1 of the FITS standard\(^3\), version 4.0.

In short, in the \( PC_{i,j} \) formalism, we only keep the linear rotation matrix in these keywords and put the scaling factor (or the pixel scale in astronomical imaging) in the \( CDELT_i \) keywords. In the \( CD_{i,j} \) formalism, we blend the scaling into the rotation into a single matrix and keep that matrix in these FITS keywords. By default, Gnuastro uses the \( PC_{i,j} \) formalism, because it greatly helps in human readability of the raw keywords and is also the default mode of WCSLIB. However, in some circumstances it may be necessary to have the keywords in the CD format; for example when you need to feed the outputs into other software that don’t follow the full FITS standard and only recognize the \( CD_{i,j} \) formalism.

- **txt**  A plain text table with white-space characters between the columns (see Section 4.7.2 [Gnuastro text table format], page 185).
- **fits-ascii**  A FITS ASCII table (see Section 4.7.1 [Recognized table formats], page 184).
- **fits-binary**  A FITS binary table (see Section 4.7.1 [Recognized table formats], page 184).

### 4.1.2.2 Processing options

Some processing steps are common to several programs, so they are defined as common options to all programs. Note that this class of common options is thus necessarily less common between all the programs than those described in Section 4.1.2.1 [Input/Output options], page 157, or Section 4.1.2.3 [Operating mode options], page 162, options. Also, if they are irrelevant for a program, these options will not display in the `--help` output of the program.

```
--minmapsize=INT
```

The minimum size (in bytes) to memory-map a processing/internal array as a file (on the non-volatile HDD/SSD), and not use the system’s RAM. Before using this option, please read Section 4.6 [Memory management], page 180. By default processing arrays will only be memory-mapped to a file when the RAM is full. With this option, you can force the memory-mapping, even when there is enough RAM. To ensure this default behavior, the pre-defined value to this option is an extremely large value (larger than any existing RAM).

Please note that using a non-volatile file (in the HDD/SSD) instead of RAM can significantly increase the program’s running time, especially on HDDs (where

---

\(^3\) [https://fits.gsfc.nasa.gov/standard40/fits_standard40aa-1e.pdf](https://fits.gsfc.nasa.gov/standard40/fits_standard40aa-1e.pdf)
read/write is slower). Also, note that the number of memory-mapped files that your kernel can support is limited. So when this option is necessary, it is best to give it values larger than 1 megabyte (--minmapsize=1000000). You can then decrease it for a specific program's invocation on a large input after you see memory issues arise (for example an error, or the program not aborting and fully consuming your memory). If you see randomly named files remaining in this directory when the program finishes normally, please send us a bug report so we address the problem, see Section 1.8 [Report a bug], page 11.

Limited number of memory-mapped files: The operating system kernels usually support a limited number of memory-mapped files. Therefore never set --minmapsize to zero or a small number of bytes (so too many files are created). If the kernel capacity is exceeded, the program will crash.

--quietmmap
Don't print any message when an array is stored in non-volatile memory (HDD/SSD) and not RAM, see the description of --minmapsize (above) for more.

-Z INT[,INT[,...]]
--tilesize=[,INT[,,...]]
The size of regular tiles for tessellation, see Section 4.8 [Tessellation], page 189. For each dimension an integer length (in units of data-elements or pixels) is necessary. If the number of input dimensions is different from the number of values given to this option, the program will stop with an error. Values must be separated by commas (,) and can also be fractions (for example 4/2). If they are fractions, the result must be an integer, otherwise an error will be printed.

-M INT[,INT[,...]]
--numchannels=INT[,INT[,...]]
The number of channels for larger input tessellation, see Section 4.8 [Tessellation], page 189. The number and types of acceptable values are similar to --tilesize. The only difference is that instead of length, the integers values given to this option represent the number of channels, not their size.

-F FLT
--remainderfrac=FLT
The fraction of remainder size along all dimensions to add to the first tile. See Section 4.8 [Tessellation], page 189, for a complete description. This option is only relevant if --tilesize is not exactly divisible by the input dataset's size in a dimension. If the remainder size is larger than this fraction (compared to --tilesize), then the remainder size will be added with one regular tile size and divided between two tiles at the start and end of the given dimension.

--workoverch
Ignore the channel borders for the high-level job of the given application. As a result, while the channel borders are respected in defining the small tiles (such that no tile will cross a channel border), the higher-level program operation will ignore them, see Section 4.8 [Tessellation], page 189.
--checktiles
Make a FITS file with the same dimensions as the input but each pixel is replaced with the ID of the tile that it is associated with. Note that the tile IDs start from 0. See Section 4.8 [Tessellation], page 189, for more on Tiling an image in Gnuastro.

--oneelempertile
When showing the tile values (for example with --checktiles, or when the program’s output is tessellated) only use one element for each tile. This can be useful when only the relative values given to each tile compared to the rest are important or need to be checked. Since the tiles usually have a large number of pixels within them the output will be much smaller, and so easier to read, write, store, or send.

Note that when the full input size in any dimension is not exactly divisible by the given --tilesize in that dimension, the edge tile(s) will have different sizes (in units of the input’s size), see --remainderfrac. But with this option, all displayed values are going to have the (same) size of one data-element. Hence, in such cases, the image proportions are going to be slightly different with this option.

If your input image is not exactly divisible by the tile size and you want one value per tile for some higher-level processing, all is not lost though. You can see how many pixels were within each tile (for example to weight the values or discard some for later processing) with Gnuastro’s Statistics (see Section 7.1 [Statistics], page 337) as shown below. The output FITS file is going to have two extensions, one with the median calculated on each tile and one with the number of elements that each tile covers. You can then use the where operator in Section 6.2 [Arithmetic], page 272, to set the values of all tiles that don’t have the regular area to a blank value.

```
$ aststatistics --median --number --ontile input.fits \ 
   --oneelempertile --output=o.fits
$ REGULAR_AREA=1600  # Check second extension of 'o.fits'.
$ astarithmetic o.fits o.fits $REGULAR_AREA ne nan where \ 
   -h1 -h2
```

Note that if input.fits also has blank values, then the median on tiles with blank values will also be ignored with the command above (which is desirable).

--inteonlyblank
When values are to be interpolated, only change the values of the blank elements, keep the non-blank elements untouched.

--interpmetric=STR
The metric to use for finding nearest neighbors. Currently it only accepts the Manhattan (or taxicab) metric with manhattan, or the radial metric with radial.

The Manhattan distance between two points is defined with $|\Delta x| + |\Delta y|$. Thus the Manhattan metric has the advantage of being fast, but at the expense of being less accurate. The radial distance is the standard definition of distance
in a Euclidean space: \( \sqrt{\Delta x^2 + \Delta y^2} \). It is accurate, but the multiplication and square root can slow down the processing.

\[ \text{--interpnumngb=INT} \]

The number of nearby non-blank neighbors to use for interpolation.

### 4.1.2.3 Operating mode options

Another group of options that are common to all the programs in Gnuastro are those to do with the general operation of the programs. The explanation for those that are not only limited to Gnuastro but are common to all GNU programs start with (GNU option).

\[ \text{--} \]

(GNU option) Stop parsing the command-line. This option can be useful in scripts or when using the shell history. Suppose you have a long list of options, and want to see if removing some of them (to read from configuration files, see Section 4.2 [Configuration files], page 169) can give a better result. If the ones you want to remove are the last ones on the command-line, you don’t have to delete them, you can just add -- before them and if you don’t get what you want, you can remove the -- and get the same initial result.

\[ \text{--usage} \]

(GNU option) Only print the options and arguments and abort. This is very useful for when you know the what the options do, and have just forgot their long/short identifiers, see Section 4.3.1 [--usage], page 172.

\[-? \]

\[ \text{--help} \]

(GNU option) Print all options with an explanation and abort. Adding this option will print all the options in their short and long formats, also displaying which ones need a value if they are called (with an = after the long format followed by a string specifying the format, see Section 4.1.1.2 [Options], page 154). A short explanation is also given for what the option is for. The program will quit immediately after the message is printed and will not do any form of processing, see Section 4.3.2 [--help], page 173.

\[-V \]

\[ \text{--version} \]

(GNU option) Print a short message, showing the full name, version, copyright information and program authors and abort. On the first line, it will print the official name (not executable name) and version number of the program. Following this is a blank line and a copyright information. The program will not run.

\[-q \]

\[ \text{--quiet} \]

Don’t report steps. All the programs in Gnuastro that have multiple major steps will report their steps for you to follow while they are operating. If you do not want to see these reports, you can call this option and only error/warning messages will be printed. If the steps are done very fast (depending on the properties of your input) disabling these reports will also decrease running time.

\[ \text{--cite} \]

Print all necessary information to cite and acknowledge Gnuastro in your published papers. With this option, the programs will print the BibTeX entry to include in your paper for Gnuastro in general, and the particular program’s paper (if that program comes with a separate paper). It will also print the
necessary acknowledgment statement to add in the respective section of your paper and it will abort. For a more complete explanation, please see Section 1.12 [Acknowledgments], page 15.

Citations and acknowledgments are vital for the continued work on Gnuastro. Gnuastro started, and is continued, based on separate research projects. So if you find any of the tools offered in Gnuastro to be useful in your research, please use the output of this command to cite and acknowledge the program (and Gnuastro) in your research paper. Thank you.

Gnuastro is still new, there is no separate paper only devoted to Gnuastro yet. Therefore currently the paper to cite for Gnuastro is the paper for NoiseChisel which is the first published paper introducing Gnuastro to the astronomical community. Upon reaching a certain point, a paper completely devoted to describing Gnuastro’s many functionalities will be published, see Section 1.6.1 [GNU Astronomy Utilities 1.0], page 8.

-p
--printparams
With this option, Gnuastro’s programs will read your command-line options and all the configuration files. If there is no problem (like a missing parameter or a value in the wrong format or range) and immediately before actually running, the programs will print the full list of option names, values and descriptions, sorted and grouped by context and abort. They will also report the version number, the date they were configured on your system and the time they were reported.

As an example, you can give your full command-line options and even the input and output file names and finally just add -p to check if all the parameters are finely set. If everything is OK, you can just run the same command (easily retrieved from the shell history, with the top arrow key) and simply remove the last two characters that showed this option.

No program will actually start its processing when this option is called. The otherwise mandatory arguments for each program (for example input image or catalog files) are no longer required when you call this option.

--config=STR
Parse STR as a configuration file name, immediately when this option is confronted (see Section 4.2 [Configuration files], page 169). The --config option can be called multiple times in one run of any Gnuastro program on the command-line or in the configuration files. In any case, it will be immediately read (before parsing the rest of the options on the command-line, or lines in a configuration file). If the given file doesn’t exist or can’t be read for any reason, the program will print a warning and continue its processing. The warning can be suppressed with --quiet.

Note that by definition, options on the command-line still take precedence over those in any configuration file, including the file(s) given to this option if they are called before it. Also see --lastconfig and --onlyversion on how this option can be used for reproducible results. You can use --checkconfig (below) to check/confirm the parsing of configuration files.
--checkconfig
Print options and their values, within the command-line or configuration files, as they are parsed (see Section 4.2.2 [Configuration file precedence], page 170). If an option has already been set, or is ignored by the program, this option will also inform you with special values like --ALREADY-SET--. Only options that are parsed after this option are printed, so to see the parsing of all input options, it is recommended to put this option immediately after the program name before any other options.

This is a very good option to confirm where the value of each option is has been defined in scenarios where there are multiple configuration files (for debugging).

-S
--setdirconf
Update the current directory configuration file for the Gnuastro program and quit. The full set of command-line and configuration file options will be parsed and options with a value will be written in the current directory configuration file for this program (see Section 4.2 [Configuration files], page 169). If the configuration file or its directory doesn’t exist, it will be created. If a configuration file exists it will be replaced (after it, and all other configuration files have been read). In any case, the program will not run.

This is the recommended method to edit/set the configuration file for all future calls to Gnuastro’s programs. It will internally check if your values are in the correct range and type and save them according to the configuration file format, see Section 4.2.1 [Configuration file format], page 169. So if there are unreasonable values to some options, the program will notify you and abort before writing the final configuration file.

When this option is called, the otherwise mandatory arguments, for example input image or catalog file(s), are no longer mandatory (since the program will not run).

-U
--setusrconf
Update the user configuration file and quit (see Section 4.2 [Configuration files], page 169). See explanation under --setdirconf for more details.

--lastconfig
This is the last configuration file that must be read. When this option is confronted in any stage of reading the options (on the command-line or in a configuration file), no other configuration file will be parsed, see Section 4.2.2 [Configuration file precedence], page 170, and Section 4.2.3 [Current directory and User wide], page 171. Like all on/off options, on the command-line, this option doesn’t take any values. But in a configuration file, it takes the values of 0 or 1, see Section 4.2.1 [Configuration file format], page 169. If it is present in a configuration file with a value of 0, then all later occurrences of this option will be ignored.

\(^4\) Alternatively, you can use your favorite text editor.
**--onlyversion=STR**

Only run the program if Gnuastro’s version is exactly equal to STR (see Section 1.6 [Version numbering], page 7). Note that it is not compared as a number, but as a string of characters, so 0, or 0.0 and 0.00 are different. If the running Gnuastro version is different, then this option will report an error and abort as soon as it is confronted on the command-line or in a configuration file. If the running Gnuastro version is the same as STR, then the program will run as if this option was not called.

This is useful if you want your results to be exactly reproducible and not mistakenly run with an updated/newer or older version of the program. Besides internal algorithmic/behavior changes in programs, the existence of options or their names might change between versions (especially in these earlier versions of Gnuastro).

Hence, when using this option (probably in a script or in a configuration file), be sure to call it before other options. The benefit is that, when the version differs, the other options won’t be parsed and you, or your collaborators/users, won’t get errors saying an option in your configuration doesn’t exist in the running version of the program.

Here is one example of how this option can be used in conjunction with the --lastconfig option. Let’s assume that you were satisfied with the results of this command: `astnoisechisel image.fits --snquant=0.95` (along with various options set in various configuration files). You can save the state of NoiseChisel and reproduce that exact result on `image.fits` later by following these steps (the extra spaces, and \, are only for easy readability, if you want to try it out, only one space between each token is enough).

```
$ echo "onlyversion X.XX" > reproducible.conf
$ echo "lastconfig 1" >> reproducible.conf
$ astnoisechisel image.fits --snquant=0.95 -P \  
    >> reproducible.conf
```

--onlyversion was available from Gnuastro 0.0, so putting it immediately at the start of a configuration file will ensure that later, you (or others using different version) won’t get a non-recognized option error in case an option was added/removed. --lastconfig will inform the installed NoiseChisel to not parse any other configuration files. This is done because we don’t want the user’s user-wide or system wide option values affecting our results. Finally, with the third command, which has a -P (short for --printparams), NoiseChisel will print all the option values visible to it (in all the configuration files) and the shell will append them to `reproduce.conf`. Hence, you don’t have to worry about remembering the (possibly) different options in the different configuration files.

 Afterwards, if you run NoiseChisel as shown below (telling it to read this configuration file with the --config option). You can be sure that there will either be an error (for version mismatch) or it will produce exactly the same result that you got before.

```
$ astnoisechisel --config=reproducible.conf
```
Chapter 4: Common program behavior

--log

Some programs can generate extra information about their outputs in a log file. When this option is called in those programs, the log file will also be printed. If the program doesn’t generate a log file, this option is ignored.

--log isn’t thread-safe: The log file usually has a fixed name. Therefore if two simultaneous calls (with --log) of a program are made in the same directory, the program will try to write to the same file. This will cause problems like unreasonable log file, undefined behavior, or a crash.

-N INT
--numthreads=INT

Use INT CPU threads when running a Gnuastro program (see Section 4.4 [Multi-threaded operations], page 175). If the value is zero (0), or this option is not given on the command-line or any configuration file, the value will be determined at run-time: the maximum number of threads available to the system when you run a Gnuastro program.

Note that multi-threaded programming is only relevant to some programs. In others, this option will be ignored.

4.1.3 Shell TAB completion (highly customized)

Under development: Gnuastro’s TAB completion in Bash already greatly improves usage of Gnuastro on the command-line, but still under development and not yet complete. If you are interested to try it out, please go ahead and activate it (as described below), we encourage this. But please have in mind that there are known issues\(^5\) and you may find new issues. If you do, please get in touch with us as described in Section 1.8 [Report a bug], page 11. TAB completion is currently only implemented in the following programs: Arithmetic, BuildProgram, ConvertType, Convolve, CosmicCalculator, Crop, Fits and Table. For progress on this task, please see Task 15799\(^6\).

Bash provides a built-in feature called *programmable completion*\(^7\) to help increase interactive workflow efficiency and minimize the number of key-strokes and the need to memorize things. It is also known as TAB completion, bash completion, auto-completion, or word completion. Completion is activated by pressing [TAB] while you are typing a command. For file arguments this is the default behavior already and you have probably used it a lot with any command-line program.

Besides this simple/default mode, Bash also enables a high level of customization features for its completion. These features have been extensively used in Gnuastro to improve your work efficiency\(^8\). For example if you are running `asttable` (which only accepts files containing a table), and you press [TAB], it will only suggest files containing tables. As

\(^5\) http://savannah.gnu.org/bugs/index.php?group=gnuastro&category_id=128  
\(^6\) https://savannah.gnu.org/task/?15799  
\(^7\) https://www.gnu.org/software/bash/manual/html_node/Programmable-Completion.html  
\(^8\) To learn how Gnuastro implements TAB completion in Bash, see Section 12.8 [Bash programmable completion], page 685.
another example, if an option needs image HDUs within a FITS file, pressing [TAB] will only suggest the image HDUs (and not other possibly existing HDUs that contain tables, or just metadata). Just note that the file name has to be already given on the command-line before reaching such options (that look into the contents of a file).

But TAB completion is not limited to file types or contents. Arguments/Options that take certain fixed string values will directly suggest those strings with TAB, and completely ignore the file structure (for example spectral line names in Section 9.1.3 [Invoking Cosmic-Calculator], page 474)! As another example, the option --numthreads option (to specify the number of threads to use by the program), will find the number of available threads on the system, and suggest the possible numbers with a TAB!

To activate Gnuastro's custom TAB completion in Bash, you need to put the following line in one of your Bash startup files (for example ~/.bashrc). If you installed Gnuastro using the steps of Section 1.1 [Quick start], page 1, you should have already done this (the command just after sudo make install). For a list of (and discussion on) Bash startup files and installation directories see Section 3.3.1.2 [Installation directory], page 139. Of course, if Gnuastro was installed in a custom location, replace the '/usr/local' part of the line below to the value that was given to --prefix during Gnuastro's configuration.

```bash
# Enable Gnuastro's TAB completion
source /usr/local/share/gnuastro/completion.bash
```

After adding the line above in a Bash startup file, TAB completion will always be activated in any new terminal. To see if it has been activated, try it out with asttable [TAB] [TAB] and astarithmetic [TAB] [TAB] in a directory that contains tables and images. The first will only suggest the files with a table, and the second, only those with an image.

### TAB completion only works with long option names:

As described above, short options are much more complex to generalize, therefore TAB completion is only available for long options. But don’t worry! TAB completion also involves option names, so if you just type --a[TAB] [TAB], you will get the list of options that start with an --a. Therefore as a side-effect of TAB completion, your commands will be far more human-readable with minimal key strokes.

#### 4.1.4 Standard input

The most common way to feed the primary/first input dataset into a program is to give its filename as an argument (discussed in Section 4.1.1.1 [Arguments], page 154). When you want to run a series of programs in sequence, this means that each will have to keep the output of each program in a separate file and re-type that file’s name in the next command. This can be very slow and frustrating (mis-typing a file’s name).

To solve the problem, the founders of Unix defined pipes to directly feed the output of one program (its “Standard output” stream) into the “standard input” of a next program. This removes the need to make temporary files between separate processes and became one of the best demonstrations of the Unix-way, or Unix philosophy.

9 In case you don’t know the installation directory of Gnuastro on your system, you can find out with this command: `which astfits | sed -e"s|/bin/astfits||"`
Every program has three streams identifying where it reads/writes non-file inputs/outputs: Standard input, Standard output, and Standard error. When a program is called alone, all three are directed to the terminal that you are using. If it needs an input, it will prompt you for one and you can type it in. Or, it prints its results in the terminal for you to see.

For example, say you have a FITS table/catalog containing the B and V band magnitudes (MAG_B and MAG_V columns) of a selection of galaxies along with many other columns. If you want to see only these two columns in your terminal, can use Gnuastro's program like below:

```
$ asttable cat.fits -c MAG_B,MAG_V
```

Through the Unix pipe mechanism, when the shell confronts the pipe character (|), it connects the standard output of the program before the pipe, to the standard input of the program after it. So it is literally a "pipe": everything that you would see printed by the first program on the command (without any pipe), is now passed to the second program (and not seen by you).

To continue the previous example, let’s say you want to see the B-V color. To do this, you can pipe Table’s output to AWK (a wonderful tool for processing things like plain text tables):

```
$ asttable cat.fits -c MAG_B,MAG_V | awk '{print $1-$2}'
```

But understanding the distribution by visually seeing all the numbers under each other is not too useful! You can therefore feed this single column information into Section 7.1 [Statistics], page 337, to give you a general feeling of the distribution with the same command:

```
$ asttable cat.fits -c MAG_B,MAG_V | awk '{print $1-$2}' | aststatistics
```

Gnuastro’s programs that accept input from standard input, only look into the Standard input stream if there is no first argument. In other words, arguments take precedence over Standard input. When no argument is provided, the programs check if the standard input stream is already full or not (output from another program is waiting to be used). If data is present in the standard input stream, it is used.

When the standard input is empty, the program will wait --stdtimeout micro-seconds for you to manually enter the first line (ending with a new-line character, or the ENTER key, see Section 4.1.2.1 [Input/Output options], page 157). If it detects the first line in this time, there is no more time limit, and you can manually write/type all the lines for as long as it takes. To inform the program that Standard input has finished, press CTRL-D after a new line. If the program doesn’t catch the first line before the time-out finishes, it will abort with an error saying that no input was provided.

**Manual input in Standard input is discarded:** Be careful that when you manually fill the Standard input, the data will be discarded once the program finishes and reproducing the result will be impossible. Therefore this form of providing input is only good for temporary tests.
4.2 Configuration files

Each program needs a certain number of parameters to run. Supplying all the necessary parameters each time you run the program is very frustrating and prone to errors. Therefore all the programs read the values for the necessary options you have not given in the command line from one of several plain text files (which you can view and edit with any text editor). These files are known as configuration files and are usually kept in a directory named etc/ according to the file system hierarchy standard\textsuperscript{10}.

The thing to have in mind is that none of the programs in Gnuastro keep any internal default value. All the values must either be stored in one of the configuration files or explicitly called in the command-line. In case the necessary parameters are not given through any of these methods, the program will print a missing option error and abort. The only exception to this is --numthreads, whose default value is determined at runtime using the number of threads available to your system, see Section 4.4 [Multi-threaded operations], page 175. Of course, you can still provide a default value for the number of threads at any of the levels below, but if you don’t, the program will not abort. Also note that through automatic output name generation, the value to the --output option is also not mandatory on the command-line or in the configuration files for all programs which don’t rely on that value as an input\textsuperscript{11}, see Section 4.9 [Automatic output], page 191.

4.2.1 Configuration file format

The configuration files for each program have the standard program executable name with a ‘.conf’ suffix. When you download the source code, you can find them in the same directory as the source code of each program, see Section 12.4 [Program source], page 677.

Any line in the configuration file whose first non-white character is a # is considered to be a comment and is ignored. An empty line is also similarly ignored. The long name of the option should be used as an identifier. The parameter name and parameter value have to be separated by any number of ‘white-space’ characters: space, tab or vertical tab. By default several space characters are used. If the value of an option has space characters (most commonly for the hdu option), then the full value can be enclosed in double quotation signs ("), similar to the example in Section 4.1.1 [Arguments and options], page 153. If it is an option without a value in the --help output (on/off option, see Section 4.1.1.2 [Options], page 154), then the value should be 1 if it is to be ‘on’ and 0 otherwise.

In each non-commented and non-blank line, any text after the first two words (option identifier and value) is ignored. If an option identifier is not recognized in the configuration file, the name of the file, the line number of the unrecognized option, and the unrecognized identifier name will be reported and the program will abort. If a parameter is repeated more than once in the configuration files, accepts only one value, and is not set on the command-line, then only the first value will be used, the rest will be ignored.

\textsuperscript{10} http://en.wikipedia.org/wiki/Filesistem_Hierarchy_Standard

\textsuperscript{11} One example of a program which uses the value given to --output as an input is ConvertType, this value specifies the type of the output through the value to --output, see Section 5.2.5 [Invoking ConvertType], page 224.
You can build or edit any of the directories and the configuration files yourself using any text editor. However, it is recommended to use the `--setdirconf` and `--setusrconf` options to set default values for the current directory or this user, see Section 4.1.2.3 [Operating mode options], page 162. With these options, the values you give will be checked before writing in the configuration file. They will also print a set of commented lines guiding the reader and will also classify the options based on their context and write them in their logical order to be more understandable.

### 4.2.2 Configuration file precedence

The option values in all the programs of Gnuastro will be filled in the following order. If an option only takes one value which is given in an earlier step, any value for that option in a later step will be ignored. Note that if the lastconfig option is specified in any step below, no other configuration files will be parsed (see Section 4.1.2.3 [Operating mode options], page 162).

1. Command-line options, for a particular run of ProgramName.
2. `.gnuastro/astprogname.conf` is parsed by ProgramName in the current directory.
3. `.gnuastro/gnuastro.conf` is parsed by all Gnuastro programs in the current directory.
4. `$HOME/.local/etc/astprogname.conf` is parsed by ProgramName in the user’s home directory (see Section 4.2.3 [Current directory and User wide], page 171).
5. `$HOME/.local/etc/gnuastro.conf` is parsed by all Gnuastro programs in the user’s home directory (see Section 4.2.3 [Current directory and User wide], page 171).
6. `prefix/etc/astprogname.conf` is parsed by ProgramName in the system-wide installation directory (see Section 4.2.4 [System wide], page 171, for `prefix`).
7. `prefix/etc/gnuastro.conf` is parsed by all Gnuastro programs in the system-wide installation directory (see Section 4.2.4 [System wide], page 171, for `prefix`).

The basic idea behind setting this progressive state of checking for parameter values is that separate users of a computer or separate folders in a user’s file system might need different values for some parameters.

**Checking the order:** You can confirm/check the order of parsing configuration files using the `--checkconfig` option with any Gnuastro program, see Section 4.1.2.3 [Operating mode options], page 162. Just be sure to place this option immediately after the program name, before any other option.

As you see above, there can also be a configuration file containing the common options in all the programs: `gnuastro.conf` (see Section 4.1.2 [Common options], page 156). If options specific to one program are specified in this file, there will be unrecognized option errors, or unexpected behavior if the option has different behavior in another program. On the other hand, there is no problem with `astprogname.conf` containing common options.\(^{12}\)

---

\(^{12}\) As an example, the `--setdirconf` and `--setusrconf` options will also write the common options they have read in their produced `astprogname.conf`. 

Manipulating the order: You can manipulate this order or add new files with the following two options which are fully described in Section 4.1.2.3 [Operating mode options], page 162:

--config Allows you to define any file to be parsed as a configuration file on the command-line or within the any other configuration file. Recall that the file given to --config is parsed immediately when this option is confronted (on the command-line or in a configuration file).

--lastconfig Allows you to stop the parsing of subsequent configuration files. Note that if this option is given in a configuration file, it will be fully read, so its position in the configuration doesn’t matter (unlike --config).

One example of benefiting from these configuration files can be this: raw telescope images usually have their main image extension in the second FITS extension, while processed FITS images usually only have one extension. If your system-wide default input extension is 0 (the first), then when you want to work with the former group of data you have to explicitly mention it to the programs every time. With this progressive state of default values to check, you can set different default values for the different directories that you would like to run Gnuastro in for your different purposes, so you won’t have to worry about this issue any more.

The same can be said about the gnuastro.conf files: by specifying a behavior in this single file, all Gnuastro programs in the respective directory, user, or system-wide steps will behave similarly. For example to keep the input’s directory when no specific output is given (see Section 4.9 [Automatic output], page 191), or to not delete an existing file if it has the same name as a given output (see Section 4.1.2.1 [Input/Output options], page 157).

4.2.3 Current directory and User wide

For the current (local) and user-wide directories, the configuration files are stored in the hidden sub-directories named .gnuastro/ and $HOME/.local/etc/ respectively. Unless you have changed it, the $HOME environment variable should point to your home directory. You can check it by running $ echo $HOME. Each time you run any of the programs in Gnuastro, this environment variable is read and placed in the above address. So if you suddenly see that your home configuration files are not being read, probably you (or some other program) has changed the value of this environment variable.

Although it might cause confusions like above, this dependence on the HOME environment variable enables you to temporarily use a different directory as your home directory. This can come in handy in complicated situations. To set the user or current directory configuration files based on your command-line input, you can use the --setdirconf or --setusrconf, see Section 4.1.2.3 [Operating mode options], page 162.

4.2.4 System wide

When Gnuastro is installed, the configuration files that are shipped with the distribution are copied into the (possibly system wide) prefix/etc/ directory. For more details on prefix, see Section 3.3.1.2 [Installation directory], page 139, (by default it is: /usr/local). This directory is the final place (with the lowest priority) that the programs in Gnuastro will check to retrieve parameter values.
If you remove an option and its value from the system wide configuration files, you either have to specify it in more immediate configuration files or set it each time in the command-line. Recall that none of the programs in Gnuastro keep any internal default values and will abort if they don’t find a value for the necessary parameters (except the number of threads and output file name). So even though you might never expect to use an optional option, it safe to have it available in this system-wide configuration file even if you don’t intend to use it frequently.

Note that in case you install Gnuastro from your distribution’s repositories, prefix will either be set to / (the root directory) or /usr, so you can find the system wide configuration variables in /etc/ or /usr/etc/. The prefix of /usr/local/ is conventionally used for programs you install from source by your self as in Section 1.1 [Quick start], page 1.

4.3 Getting help

Probably the first time you read this book, it is either in the PDF or HTML formats. These two formats are very convenient for when you are not actually working, but when you are only reading. Later on, when you start to use the programs and you are deep in the middle of your work, some of the details will inevitably be forgotten. Going to find the PDF file (printed or digital) or the HTML web page is a major distraction.

GNU software have a very unique set of tools for aiding your memory on the command-line, where you are working, depending how much of it you need to remember. In the past, such command-line help was known as “online” help, because they were literally provided to you ‘on’ the command ‘line’. However, nowadays the word “online” refers to something on the internet, so that term will not be used. With this type of help, you can resume your exciting research without taking your hands off the keyboard.

Another major advantage of such command-line based help routines is that they are installed with the software in your computer, therefore they are always in sync with the executable you are actually running. Three of them are actually part of the executable. You don’t have to worry about the version of the book or program. If you rely on external help (a PDF in your personal print or digital archive or HTML from the official web page) you have to check to see if their versions fit with your installed program.

If you only need to remember the short or long names of the options, --usage is advised. If it is what the options do, then --help is a great tool. Man pages are also provided for those who are use to this older system of documentation. This full book is also available to you on the command-line in Info format. If none of these seems to resolve the problems, there is a mailing list which enables you to get in touch with experienced Gnuastro users. In the subsections below each of these methods are reviewed.

4.3.1 --usage

If you give this option, the program will not run. It will only print a very concise message showing the options and arguments. Everything within square brackets ([ ]) is optional. For example here are the first and last two lines of Crop’s --usage is shown:

$$\texttt{astcrop --usage}$$

....

Chapter 4: Common program behavior

```bash
[--setusrconf] [--usage] [--version] [--wcsmode]
[ASCIIcatalog] FITSimage(s).fits
```

There are no explanations on the options, just their short and long names shown separately. After the program name, the short format of all the options that don’t require a value (on/off options) is displayed. Those that do require a value then follow in separate brackets, each displaying the format of the input they want, see Section 4.1.1.2 [Options], page 154. Since all options are optional, they are shown in square brackets, but arguments can also be optional. For example in this example, a catalog name is optional and is only required in some modes. This is a standard method of displaying optional arguments for all GNU software.

### 4.3.2 `--help`

If the command-line includes this option, the program will not be run. It will print a complete list of all available options along with a short explanation. The options are also grouped by their context. Within each context, the options are sorted alphabetically. Since the options are shown in detail afterwards, the first line of the `--help` output shows the arguments and if they are optional or not, similar to Section 4.3.1 `--usage`, page 172.

In the `--help` output of all programs in Gnuastro, the options for each program are classified based on context. The first two contexts are always options to do with the input and output respectively. For example input image extensions or supplementary input files for the inputs. The last class of options is also fixed in all of Gnuastro, it shows operating mode options. Most of these options are already explained in Section 4.1.2.3 [Operating mode options], page 162.

The help message will sometimes be longer than the vertical size of your terminal. If you are using a graphical user interface terminal emulator, you can scroll the terminal with your mouse, but we promised no mice distractions! So here are some suggestions:

- **Shift + PageUP** to scroll up and **Shift + PageDown** to scroll down. For most help output this should be enough. The problem is that it is limited by the number of lines that your terminal keeps in memory and that you can’t scroll by lines, only by whole screens.

- **Pipe to `less`.** A pipe is a form of shell re-direction. The `less` tool in Unix-like systems was made exactly for such outputs of any length. You can pipe (|) the output of any program that is longer than the screen to it and then you can scroll through (up and down) with its many tools. For example:

  ```bash
  $ astnoisechisel --help | less
  ``

  Once you have gone through the text, you can quit `less` by pressing the `q` key.

- **Redirect to a file.** This is a less convenient way, because you will then have to open the file in a text editor! You can do this with the shell redirection tool (`>`, `>`):

  ```bash
  $ astnoisechisel --help > filename.txt
  ``

  In case you have a special keyword you are looking for in the help, you don’t have to go through the full list. GNU Grep is made for this job. For example if you only want the list of options whose `--help` output contains the word “axis” in Crop, you can run the following command:

  ```bash
  $ astcrop --help | grep axis
  ```
If the output of this option does not fit nicely within the confines of your terminal, GNU does enable you to customize its output through the environment variable `ARGP_HELP_FMT`, you can set various parameters which specify the formatting of the help messages. For example if your terminals are wider than 70 spaces (say 100) and you feel there is too much empty space between the long options and the short explanation, you can change these formats by giving values to this environment variable before running the program with the `--help` output. You can define this environment variable in this manner:

```bash
$ export ARGP_HELP_FMT=rmargin=100, opt-doc-col=20
```

This will affect all GNU programs using GNU C library's `argp.h` facilities as long as the environment variable is in memory. You can see the full list of these formatting parameters in the “Argp User Customization” part of the GNU C library manual. If you are more comfortable to read the `--help` outputs of all GNU software in your customized format, you can add your customization (similar to the line above, without the `$` sign) to your `~/.bashrc` file. This is a standard option for all GNU software.

### 4.3.3 Man pages

Man pages were the Unix method of providing command-line documentation to a program. With GNU Info, see Section 4.3.4 [Info], page 174, the usage of this method of documentation is highly discouraged. This is because Info provides a much more easier to navigate and read environment.

However, some operating systems require a man page for packages that are installed and some people are still used to this method of command line help. So the programs in Gnaastro also have Man pages which are automatically generated from the outputs of `--version` and `--help` using the GNU help2man program. So if you run

```bash
$ man programname
```

You will be provided with a man page listing the options in the standard manner.

### 4.3.4 Info

Info is the standard documentation format for all GNU software. It is a very useful command-line document viewing format, fully equipped with links between the various pages and menus and search capabilities. As explained before, the best thing about it is that it is available for you the moment you need to refresh your memory on any command-line tool in the middle of your work without having to take your hands off the keyboard. This complete book is available in Info format and can be accessed from anywhere on the command-line.

To open the Info format of any installed programs or library on your system which has an Info format book, you can simply run the command below (change `executablename` to the executable name of the program or library):

```bash
$ info executablename
```

In case you are not already familiar with it, run `$ info info`. It does a fantastic job in explaining all its capabilities its self. It is very short and you will become sufficiently fluent in about half an hour. Since all GNU software documentation is also provided in Info, your whole GNU/Linux life will significantly improve.

Once you’ve become an efficient navigator in Info, you can go to any part of this book or any other GNU software or library manual, no matter how long it is, in a matter of seconds.
Chapter 4: Common program behavior

It also blends nicely with GNU Emacs (a text editor) and you can search manuals while you are writing your document or programs without taking your hands off the keyboard, this is most useful for libraries like the GNU C library. To be able to access all the Info manuals installed in your GNU/Linux within Emacs, type Ctrl-H + i.

To see this whole book from the beginning in Info, you can run

$ info gnuastro

If you run Info with the particular program executable name, for example astcrop or astnoisechisel:

$ info astprogramname

you will be taken to the section titled “Invoking ProgramName” which explains the inputs and outputs along with the command-line options for that program. Finally, if you run Info with the official program name, for example Crop or NoiseChisel:

$ info ProgramName

you will be taken to the top section which introduces the program. Note that in all cases, Info is not case sensitive.

4.3.5 help-gnuastro mailing list

Gnuastro maintains the help-gnuastro mailing list for users to ask any questions related to Gnuastro. The experienced Gnuastro users and some of its developers are subscribed to this mailing list and your email will be sent to them immediately. However, when contacting this mailing list please have in mind that they are possibly very busy and might not be able to answer immediately.

To ask a question from this mailing list, send a mail to help-gnuastro@gnu.org. Anyone can view the mailing list archives at http://lists.gnu.org/archive/html/help-gnuastro/. It is best that before sending a mail, you search the archives to see if anyone has asked a question similar to yours. If you want to make a suggestion or report a bug, please don’t send a mail to this mailing list. We have other mailing lists and tools for those purposes, see Section 1.8 [Report a bug], page 11, or Section 1.9 [Suggest new feature], page 13.

4.4 Multi-threaded operations

Some of the programs benefit significantly when you use all the threads your computer’s CPU has to offer to your operating system. The number of threads available can be larger than the number of physical (hardware) cores in the CPU (also known as Simultaneous multithreading). For example, in Intel’s CPUs (those that implement its Hyper-threading technology) the number of threads is usually double the number of physical cores in your CPU. On a GNU/Linux system, the number of threads available can be found with the command $ nproc command (part of GNU Coreutils).

Gnuastro’s programs can find the number of threads available to your system internally at run-time (when you execute the program). However, if a value is given to the --numthreads option, the given number will be used, see Section 4.1.2.3 [Operating mode options], page 162, and Section 4.2 [Configuration files], page 169, for ways to use this option. Thus --numthreads is the only common option in Gnuastro’s programs with a value that doesn’t have to be specified anywhere on the command-line or in the configuration files.
4.4.1 A note on threads

Spinning off threads is not necessarily the most efficient way to run an application. Creating a new thread isn’t a cheap operation for the operating system. It is most useful when the input data are fixed and you want the same operation to be done on parts of it. For example one input image to Crop and multiple crops from various parts of it. In this fashion, the image is loaded into memory once, all the crops are divided between the number of threads internally and each thread cuts out those parts which are assigned to it from the same image. On the other hand, if you have multiple images and you want to crop the same region(s) out of all of them, it is much more efficient to set `--numthreads=1` (so no threads spin off) and run Crop multiple times simultaneously, see Section 4.4.2 [How to run simultaneous operations], page 176.

You can check the boost in speed by first running a program on one of the data sets with the maximum number of threads and another time (with everything else the same) and only using one thread. You will notice that the wall-clock time (reported by most programs at their end) in the former is longer than the latter divided by number of physical CPU cores (not threads) available to your operating system. Asymptotically these two times can be equal (most of the time they aren’t). So limiting the programs to use only one thread and running them independently on the number of available threads will be more efficient.

Note that the operating system keeps a cache of recently processed data, so usually, the second time you process an identical data set (independent of the number of threads used), you will get faster results. In order to make an unbiased comparison, you have to first clean the system’s cache with the following command between the two runs.

```
$ sync; echo 3 | sudo tee /proc/sys/vm/drop_caches
```

**SUMMARY: Should I use multiple threads?** Depends:
- If you only have **one** data set (image in most cases!), then yes, the more threads you use (with a maximum of the number of threads available to your OS) the faster you will get your results.
- If you want to run the same operation on **multiple** data sets, it is best to set the number of threads to 1 and use Make, or GNU Parallel, as explained in Section 4.4.2 [How to run simultaneous operations], page 176.

4.4.2 How to run simultaneous operations

There are two\(^{13}\) approaches to simultaneously execute a program: using GNU Parallel or Make (GNU Make is the most common implementation). The first is very useful when you only want to do one job multiple times and want to get back to your work without actually keeping the command you ran. The second is usually for more important operations, with lots of dependencies between the different products (for example a full scientific research).

\(^{13}\) A third way would be to open multiple terminal emulator windows in your GUI, type the commands separately on each and press Enter once on each terminal, but this is far too frustrating, tedious and prone to errors. It’s therefore not a realistic solution when tens, hundreds or thousands of operations (your research targets, multiplied by the operations you do on each) are to be done.
GNU Parallel

When you only want to run multiple instances of a command on different threads and get on with the rest of your work, the best method is to use GNU parallel. Surprisingly GNU Parallel is one of the few GNU packages that has no Info documentation but only a Man page, see Section 4.3.4 [Info], page 174. So to see the documentation after installing it please run

$ man parallel

As an example, let’s assume we want to crop a region fixed on the pixels (500, 600) with the default width from all the FITS images in the ./data directory ending with sci.fits to the current directory. To do this, you can run:

$ parallel astcrop --numthreads=1 --xc=500 --yc=600 ::: 
./data/*sci.fits

GNU Parallel can help in many more conditions, this is one of the simplest, see the man page for lots of other examples. For absolute beginners: the backslash (\) is only a line breaker to fit nicely in the page. If you type the whole command in one line, you should remove it.

Make

Make is a program for building “targets” (e.g., files) using “recipes” (a set of operations) when their known “prerequisites” (other files) have been updated. It elegantly allows you to define dependency structures for building your final output and updating it efficiently when the inputs change. It is the most common infra-structure to build software today.

Scientific research methodology is very similar to software development: you start by testing a hypothesis on a small sample of objects/targets with a simple set of steps. As you are able to get promising results, you improve the method and use it on a larger, more general, sample. In the process, you will confront many issues that have to be corrected (bugs in software development jargon).

Make is a wonderful tool to manage this style of development.

Besides the raw data analysis pipeline, Make has been used to for producing reproducible papers, for example see the reproduction pipeline (https://gitlab.com/makhlaghi/NoiseChisel-paper) of the paper introducing Section 7.2 [NoiseChisel], page 362, (one of Gnuastro’s programs). In fact the NoiseChisel paper’s Make-based workflow was the foundation of a parallel project called Maneage (http://maneage.org) (Managing data lineage): http://maneage.org that is described more fully in Akhlaghi et al. 2021 (https://arxiv.org/abs/2006.03018). Therefore, it is a very useful tool for complex scientific workflows.

GNU Make\(^\text{14}\) is the most common implementation which (similar to nearly all GNU programs, comes with a wonderful manual\(^\text{15}\)). Make is very basic and simple, and thus the manual is short (the most important parts are in the first roughly 100 pages) and easy to read/understand.

\(^{14}\)https://www.gnu.org/software/make/
\(^{15}\)https://www.gnu.org/software/make/manual/
Make comes with a `--jobs (-j)` option which allows you to specify the maximum number of jobs that can be done simultaneously. For example if you have 8 threads available to your operating system. You can run:

```bash
$ make -j8
```

With this command, Make will process your `Makefile` and create all the targets (can be thousands of FITS images for example) simultaneously on 8 threads, while fully respecting their dependencies (only building a file/target when its prerequisites are successfully built). Make is thus strongly recommended for managing scientific research where robustness, archiving, reproducibility and speed\textsuperscript{16} are important.

### 4.5 Numeric data types

At the lowest level, the computer stores everything in terms of 1 or 0. For example, each program in Gnuastro, or each astronomical image you take with the telescope is actually a string of millions of these zeros and ones. The space required to keep a zero or one is the smallest unit of storage, and is known as a bit. However, understanding and manipulating this string of bits is extremely hard for most people. Therefore, different standards are defined to package the bits into separate types with a fixed interpretation of the bits in each package.

To store numbers, the most basic standard/type is for integers ($..., -2, -1, 0, 1, 2, ...$). The common integer types are 8, 16, 32, and 64 bits wide (more bits will give larger limits). Each bit corresponds to a power of 2 and they are summed to create the final number. In the integer types, for each width there are two standards for reading the bits: signed and unsigned. In the ‘signed’ convention, one bit is reserved for the sign (stating that the integer is positive or negative). The ‘unsigned’ integers use that bit in the actual number and thus contain only positive numbers (starting from zero).

Therefore, at the same number of bits, both signed and unsigned integers can allow the same number of integers, but the positive limit of the `unsigned` types is double their `signed` counterparts with the same width (at the expense of not having negative numbers). When the context of your work doesn’t involve negative numbers (for example counting, where negative is not defined), it is best to use the `unsigned` types. For the full numerical range of all integer types, see below.

Another standard of converting a given number of bits to numbers is the floating point standard, this standard can \textit{approximately} store any real number with a given precision. There are two common floating point types: 32-bit and 64-bit, for single and double precision floating point numbers respectively. The former is sufficient for data with less than 8 significant decimal digits (most astronomical data), while the latter is good for less than 16 significant decimal digits. The representation of real numbers as bits is much more complex than integers. If you are interested to learn more about it, you can start with the Wikipedia article (https://en.wikipedia.org/wiki/Floating_point).

\textsuperscript{16} Besides its multi-threaded capabilities, Make will only re-build those targets that depend on a change you have made, not the whole work. For example, if you have set the prerequisites properly, you can easily test the changing of a parameter on your paper’s results without having to re-do everything (which is much faster). This allows you to be much more productive in easily checking various ideas/assumptions of the different stages of your research and thus produce a more robust result for your exciting science.
Practically, you can use Gnuastro’s Arithmetic program to convert/change the type of an image/datacube (see Section 6.2 [Arithmetic], page 272), or Gnuastro Table program to convert a table column’s data type (see Section 5.3.1 [Column arithmetic], page 230). Conversion of a dataset’s type is necessary in some contexts. For example the program/library, that you intend to feed the data into, only accepts floating point values, but you have an integer image/column. Another situation that conversion can be helpful is when you know that your data only has values that fit within \texttt{int8} or \texttt{uint16}. However it is currently formatted in the \texttt{float64} type.

The important thing to consider is that operations involving wider, floating point, or signed types can be significantly slower than smaller-width, integer, or unsigned types respectively. Note that besides speed, a wider type also requires much more storage space (by 4 or 8 times). Therefore, when you confront such situations that can be optimized and want to store/archive/transfer the data, it is best to use the most efficient type. For example if your dataset (image or table column) only has positive integers less than $65535$, store it as an unsigned 16-bit integer for faster processing, faster transfer, and less storage space.

The short and long names for the recognized numeric data types in Gnuastro are listed below. Both short and long names can be used when you want to specify a type. For example, as a value to the common option --type (see Section 4.1.2.1 [Input/Output options], page 157), or in the information comment lines of Section 4.7.2 [Gnuastro text table format], page 185. The ranges listed below are inclusive.

\begin{itemize}
\item \texttt{u8} \quad 8-bit unsigned integers, range: 
\[0 \text{ to } 2^8 - 1\] or \[0 \text{ to } 255\].
\item \texttt{i8} \quad 8-bit signed integers, range:
\[-2^7 \text{ to } 2^7 - 1\] or \[-128 \text{ to } 127\].
\item \texttt{u16} \quad 16-bit unsigned integers, range:
\[0 \text{ to } 2^{16} - 1\] or \[0 \text{ to } 65535\].
\item \texttt{i16} \quad 16-bit signed integers, range:
\[-2^{15} \text{ to } 2^{15} - 1\] or \[-32768 \text{ to } 32767\].
\item \texttt{u32} \quad 32-bit unsigned integers, range:
\[0 \text{ to } 2^{32} - 1\] or \[0 \text{ to } 4294967295\].
\item \texttt{i32} \quad 32-bit signed integers, range:
\[-2^{31} \text{ to } 2^{31} - 1\] or \[-2147483648 \text{ to } 2147483647\].
\item \texttt{u64} \quad 64-bit unsigned integers, range
\[0 \text{ to } 2^{64} - 1\] or \[0 \text{ to } 18446744073709551615\].
\item \texttt{i64} \quad 64-bit signed integers, range:
\[-2^{63} \text{ to } 2^{63} - 1\] or \[-9223372036854775808 \text{ to } 9223372036854775807\].
\end{itemize}
f32
float32  32-bit (single-precision) floating point types. The maximum (minimum is its negative) possible value is $3.402823 \times 10^{38}$. Single-precision floating points can accurately represent a floating point number up to $\sim 7.2$ significant decimals. Given the heavy noise in astronomical data, this is usually more than sufficient for storing results.

f64
float64 64-bit (double-precision) floating point types. The maximum (minimum is its negative) possible value is $\sim 10^{308}$. Double-precision floating points can accurately represent a floating point number $\sim 15.9$ significant decimals. This is usually good for processing (mixing) the data internally, for example a sum of single precision data (and later storing the result as float32).

Some file formats don’t recognize all types. For example the FITS standard (see Section 5.1 [Fits], page 194) does not define uint64 in binary tables or images. When a type is not acceptable for output into a given file format, the respective Gnuastro program or library will let you know and abort. On the command-line, you can convert the numerical type of an image, or table column into another type with Section 6.2 [Arithmetic], page 272, or Section 5.3 [Table], page 230, respectively. If you are writing your own program, you can use the gal_data_copy_to_new_type() function in Gnuastro’s library, see Section 11.3.6.4 [Copying datasets], page 549.

4.6 Memory management

In this section we’ll review how Gnuastro manages your input data in your system’s memory. Knowing this can help you optimize your usage (in speed and memory consumption) when the data volume is large and approaches, or exceeds, your available RAM (usually in various calls to multiple programs simultaneously). But before diving into the details, let’s have a short basic introduction to memory in general and in particular the types of memory most relevant to this discussion.

Input datasets (that are later fed into programs for analysis) are commonly first stored in non-volatile memory. This is a type of memory that doesn’t need a constant power supply to keep the data and is therefore primarily aimed for long-term storage, like HDDs or SSDs. So data in this type of storage is preserved when you turn off your computer. But by its nature, non-volatile memory is much slower, in reading or writing, than the speeds that CPUs can process the data. Thus relying on this type of memory alone would create a bad bottleneck in the input/output (I/O) phase of any processing.

The first step to decrease this bottleneck is to have a faster storage space, but with a much limited storage volume. For this type of storage, computers have a Random Access Memory (or RAM). RAM is classified as a volatile memory because it needs a constant flow of electricity to keep the information. In other words, the moment power is cut-off, all the stored information in your RAM is gone (hence the “volatile” name). But thanks to that constant supply of power, it can access any random address with equal (and very high!) speed.
Hence, the general/simplistic way that programs deal with memory is the following (this is general to almost all programs, not just Gnuastro’s): 1) Load/copy the input data from the non-volatile memory into RAM. 2) Use the copy of the data in RAM as input for all the internal processing as well as the intermediate data that is necessary during the processing. 3) Finally, when the analysis is complete, write the final output data back into non-volatile memory, and free/delete all the used space in the RAM (the initial copy and all the intermediate data). Usually the RAM is most important for the data of the intermediate steps (that you never see as a user of a program!).

When the input dataset(s) to a program are small (compared to the available space in your system’s RAM at the moment it is run) Gnuastro’s programs and libraries follow the standard series of steps above. The only exception is that deleting the intermediate data is not only done at the end of the program. As soon as an intermediate dataset is no longer necessary for the next internal steps, the space it occupied is deleted/freed. This allows Gnuastro programs to minimize their usage of your system’s RAM over the full running time.

The situation gets complicated when the datasets are large (compared to your available RAM when the program is run). For example if a dataset is half the size of your system’s available RAM, and the program’s internal analysis needs three or more intermediately processed copies of it at one moment in its analysis. There won’t be enough RAM to keep those higher-level intermediate data. In such cases, programs that don’t do any memory management will crash. But fortunately Gnuastro’s programs do have a memory management plans for such situations.

When the necessary amount of space for an intermediate dataset cannot be allocated in the RAM, Gnuastro’s programs will not use the RAM at all. They will use the “memory-mapped file” concept in modern operating systems to create a randomly-named file in your non-volatile memory and use that instead of the RAM. That file will have the exact size (in bytes) of that intermediate dataset. Any time the program needs that intermediate dataset, the operating system will directly go to that file, and bypass your RAM. As soon as that file is no longer necessary for the analysis, it will be deleted. But as mentioned above, non-volatile memory has much slower I/O speed than the RAM. Hence in such situations, the programs will become noticeably slower (sometimes by factors of 10 times slower, depending on your non-volatile memory speed).

Because of the drop in I/O speed (and thus the speed of your running program), the moment that any to-be-allocated dataset is memory-mapped, Gnuastro’s programs and libraries will notify you with a descriptive statement like below (can happen in any phase of their analysis). It shows the location of the memory-mapped file, its size, complemented with a small description of the cause, a pointer to this section of the book for more information on how to deal with it (if necessary), and what to do to suppress it.

```
astarithmetic: ./gnuastro_mmap/Fu7Dhs: temporary memory-mapped file (XXXXXXXXXX bytes) created for intermediate data that is not stored in RAM (see the "Memory management" section of Gnuastro's manual for optimizing your project's memory management, and thus speed). To disable this warning, please use the option '--quiet-mmap'
```

Finally, when the intermediate dataset is no longer necessary, the program will automatically delete it and notify you with a statement like this:
To disable these messages, you can run the program with \texttt{--quietmmap}, or set the \texttt{quietmmap} variable in the allocating library function to be non-zero.

An important component of these messages is the name of the memory-mapped file. Knowing that the file has been deleted is important for the user if the program crashes for any reason: internally (for example a parameter is given wrongly) or externally (for example you mistakenly kill the running job). In the event of a crash, the memory-mapped files will not be deleted and you have to manually delete them because they are usually large and they may soon fill your full storage if not deleted in a long time due to successive crashes.

This brings us to managing the memory-mapped files in your non-volatile memory. In other words: knowing where they are saved, or intentionally placing them in different places of your file system, or deleting them when necessary. As the examples above show, memory-mapped files are stored in a sub-directory of the running directory called \texttt{gnuastro_mmap}. If this directory doesn’t exist, Gnuastro will automatically create it when memory mapping becomes necessary. Alternatively, it may happen that the \texttt{gnuastro_mmap} sub-directory exists and isn’t writable, or it can’t be created. In such cases, the memory-mapped file for each dataset will be created in the running directory with a \texttt{gnuastro_mmap} prefix.

Therefore one easy way to delete all memory-mapped files in case of a crash, is to delete everything within the sub-directory (first command below), or all files stating with this prefix:

\begin{verbatim}
rm -f gnuastro_mmap/*
rm -f gnuastro_mmap_*
\end{verbatim}

A much more common issue when dealing with memory-mapped files is their location. For example, you may be running a program in a partition that is hosted by an HDD. But you also have another partition on an SSD (which has much faster I/O). So you want your memory-mapped files to be created in the SSD to speed up your processing. In this scenario, you want your project source directory to only contain your plain-text scripts and you want your project’s built products (even the temporary memory-mapped files) to be built in a different location because they are large; thus I/O speed becomes important.

To host the memory-mapped files in another location (with fast I/O), you can set (\texttt{gnuastro_mmap}) to be a symbolic link to it. For example, let’s assume you want your memory-mapped files to be stored in \texttt{/path/to/dir/for/mmap}. All you have to do is to run the following command before your Gnuastro analysis command(s).

\begin{verbatim}
ln -s /path/to/dir/for/mmap gnuastro_mmap
\end{verbatim}

The programs will delete a memory-mapped file when it is no longer needed, but they won’t delete the \texttt{gnuastro_mmap} directory that hosts them. So if your project involves many Gnuastro programs (possibly called in parallel) and you want your memory-mapped files to be in a different location, you just have to make the symbolic link above once at the start, and all the programs will use it if necessary.

Another memory-management scenario that may happen is this: you don’t want a Gnuastro program to allocate internal datasets in the RAM at all. For example the speed of your Gnuastro-related project doesn’t matter at that moment, and you have higher-priority jobs that are being run at the same time which need to have RAM available. In such cases, you can use the \texttt{--minmapsize} option that is available in all Gnuastro programs (see Section 4.1.2.2 [Processing options], page 159). Any intermediate dataset that has a size
larger than the value of this option will be memory-mapped, even if there is space available in your RAM. For example if you want any dataset larger than 100 megabytes to be memory-mapped, use \texttt{--minmapsize=100000000} (8 zeros!).

You shouldn’t set the value of \texttt{--minmapsize} to be too small, otherwise even small intermediate values (that are usually very numerous) in the program will be memory-mapped. However the kernel can only host a limited number of memory-mapped files at every moment (by all running programs combined). For example in the default\footnote{If you need to host more memory-mapped files at one moment, you need to build your own customized Linux kernel.} Linux kernel on GNU/Linux operating systems this limit is roughly 64000. If the total number of memory-mapped files exceeds this number, all the programs using them will crash. Gnuastro’s programs will warn you if your given value is too small and may cause a problem later.

Actually, the default behavior for Gnuastro’s programs (to only use memory-mapped files when there isn’t enough RAM) is a side-effect of \texttt{--minmapsize}. The pre-defined value to this option is an extremely large value in the lowest-level Gnuastro configuration file (the installed \texttt{gnuastro.conf} described in Section 4.2.2 [Configuration file precedence], page 170). This value is larger than the largest possible available RAM. You can check by running any Gnuastro program with a \texttt{-P} option. Because no dataset will be larger than this, by default the programs will first attempt to use the RAM for temporary storage. But if writing in the RAM fails (for any reason, mainly due to lack of available space), then a memory-mapped file will be created.

### 4.7 Tables

“A table is a collection of related data held in a structured format within a database. It consists of columns, and rows.” (from Wikipedia). Each column in the table contains the values of one property and each row is a collection of properties (columns) for one target object. For example, let’s assume you have just ran \texttt{MakeCatalog} (see Section 7.4 \texttt{[MakeCatalog]}, page 392) on an image to measure some properties for the labeled regions (which might be detected galaxies for example) in the image. For each labeled region (detected galaxy), there will be a row which groups its measured properties as columns, one column for each property. One such property can be the object’s magnitude, which is the sum of pixels with that label, or its center can be defined as the light-weighted average value of those pixels. Many such properties can be derived from the raw pixel values and their position, see Section 7.4.6 \texttt{[Invoking MakeCatalog]}, page 408, for a long list.

As a summary, for each labeled region (or, galaxy) we have one row and for each measured property we have one column. This high-level structure is usually the first step for higher-level analysis, for example finding the stellar mass or photometric redshift from magnitudes in multiple colors. Thus, tables are not just outputs of programs, in fact it is much more common for tables to be inputs of programs. For example, to make a mock galaxy image, you need to feed in the properties of each galaxy into Section 8.1 \texttt{[MakeProfiles]}, page 440, for it do the inverse of the process above and make a simulated image from a catalog, see Section 2.1 \texttt{[Sufi simulates a detection]}, page 18. In other cases, you can feed a table into Section 6.1 \texttt{[Crop]}, page 259, and it will crop out regions centered on the positions within the table, see Section 2.2.19 \texttt{[Finding reddest clumps and visual inspection]}, page 68.
So to end this relatively long introduction, tables play a very important role in astronomy, or generally all branches of data analysis.

In Section 4.7.1 [Recognized table formats], page 184, the currently recognized table formats in Gnuastro are discussed. You can use any of these tables as input or ask for them to be built as output. The most common type of table format is a simple plain text file with each row on one line and columns separated by white space characters, this format is easy to read/write by eye/hand. To give it the full functionality of more specific table types like the FITS tables, Gnuastro has a special convention which you can use to give each column a name, type, unit, and comments, while still being readable by other plain text table readers. This convention is described in Section 4.7.2 [Gnuastro text table format], page 185.

When tables are input to a program, the program reading it needs to know which column(s) it should use for its desired purposes. Gnuastro’s programs all follow a similar convention, on the way you can select columns in a table. They are thoroughly discussed in Section 4.7.3 [Selecting table columns], page 188.

4.7.1 Recognized table formats

The list of table formats that Gnuastro can currently read from and write to are described below. Each has their own advantage and disadvantages, so a short review of the format is also provided to help you make the best choice based on how you want to define your input tables or later use your output tables.

Plain text table

This is the most basic and simplest way to create, view, or edit the table by hand on a text editor. The other formats described below are less eye-friendly and have a more formal structure (for easier computer readability). It is fully described in Section 4.7.2 [Gnuastro text table format], page 185.

FITS ASCII tables

The FITS ASCII table extension is fully in ASCII encoding and thus easily readable on any text editor (assuming it is the only extension in the FITS file). If the FITS file also contains binary extensions (for example an image or binary table extensions), then there will be many hard to print characters. The FITS ASCII format doesn’t have new line characters to separate rows. In the FITS ASCII table standard, each row is defined as a fixed number of characters (value to the NAXIS1 keyword), so to visually inspect it properly, you would have to adjust your text editor’s width to this value. All columns start at given character positions and have a fixed width (number of characters).

Numbers in a FITS ASCII table are printed into ASCII format, they are not in binary (that the CPU uses). Hence, they can take a larger space in memory, lose their precision, and take longer to read into memory. If you are dealing with integer type columns (see Section 4.5 [Numeric data types], page 178), another issue with FITS ASCII tables is that the type information for the column will be lost (there is only one integer type in FITS ASCII tables). One problem with the binary format on the other hand is that it isn’t portable (different CPUs/compilers) have different standards for translating the zeros and ones. But since ASCII characters are defined on a byte and are well rec-
gnized, they are better for portability on those various systems. Gnuastro’s plain text table format described below is much more portable and easier to read/write/interpret by humans manually.

Generally, as the name implies, this format is useful for when your table mainly contains ASCII columns (for example file names, or descriptions). They can be useful when you need to include columns with structured ASCII information along with other extensions in one FITS file. In such cases, you can also consider header keywords (see Section 5.1 [Fits], page 194).

FITS binary tables

The FITS binary table is the FITS standard’s solution to the issues discussed with keeping numbers in ASCII format as described under the FITS ASCII table title above. Only columns defined as a string type (a string of ASCII characters) are readable in a text editor. The portability problem with binary formats discussed above is mostly solved thanks to the portability of CFITSIO (see Section 3.1.1.2 [CFITSIO], page 120) and the very long history of the FITS format which has been widely used since the 1970s.

In the case of most numbers, storing them in binary format is more memory efficient than ASCII format. For example, to store \( -25.72034 \) in ASCII format, you need 9 bytes/characters. But if you keep this same number (to the approximate precision possible) as a 4-byte (32-bit) floating point number, you can keep/transmit it with less than half the amount of memory. When catalogs contain thousands/millions of rows in tens/hundreds of columns, this can lead to significant improvements in memory/band-width usage. Moreover, since the CPU does its operations in the binary formats, reading the table in and writing it out is also much faster than an ASCII table.

When you are dealing with integer numbers, the compression ratio can be even better, for example if you know all of the values in a column are positive and less than 255, you can use the unsigned char type which only takes one byte! If they are between \(-128\) and \(127\), then you can use the (signed) char type. So if you are thoughtful about the limits of your integer columns, you can greatly reduce the size of your file and also the speed at which it is read/written. This can be very useful when sharing your results with collaborators or publishing them. To decrease the file size even more you can name your output as ending in \( .fits.gz \) so it is also compressed after creation. Just note that compression/decompressing is CPU intensive and can slow down the writing/reading of the file.

Fortunately the FITS Binary table format also accepts ASCII strings as column types (along with the various numerical types). So your dataset can also contain non-numerical columns.

4.7.2 Gnuastro text table format

Plain text files are the most generic, portable, and easiest way to (manually) create, (visually) inspect, or (manually) edit a table. In this format, the ending of a row is defined by the new-line character (a line on a text editor). So when you view it on a text editor, every row will occupy one line. The delimiters (or characters separating the columns) are white
space characters (space, horizontal tab, vertical tab) and a comma (,). The only further requirement is that all rows/lines must have the same number of columns.

The columns don’t have to be exactly under each other and the rows can be arbitrarily long with different lengths. For example the following contents in a file would be interpreted as a table with 4 columns and 2 rows, with each element interpreted as a double type (see Section 4.5 [Numeric data types], page 178).

<table>
<thead>
<tr>
<th></th>
<th>2.234948</th>
<th>128</th>
<th>39.8923e8</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>, 4.454</td>
<td>792</td>
<td>72.98348e7</td>
</tr>
</tbody>
</table>

However, the example above has no other information about the columns (it is just raw data, with no meta-data). To use this table, you have to remember what the numbers in each column represent. Also, when you want to select columns, you have to count their position within the table. This can become frustrating and prone to bad errors (getting the columns wrong) especially as the number of columns increase. It is also bad for sending to a colleague, because they will find it hard to remember/use the columns properly.

To solve these problems in Gnuastro's programs/libraries you aren’t limited to using the column’s number, see Section 4.7.3 [Selecting table columns], page 188. If the columns have names, units, or comments you can also select your columns based on searches/matches in these fields, for example see Section 5.3 [Table], page 230. Also, in this manner, you can’t guide the program reading the table on how to read the numbers. As an example, the first and third columns above can be read as integer types: the first column might be an ID and the third can be the number of pixels an object occupies in an image. So there is no need to read these to columns as a double type (which takes more memory, and is slower).

In the bare-minimum example above, you also can’t use strings of characters, for example the names of filters, or some other identifier that includes non-numerical characters. In the absence of any information, only numbers can be read robustly. Assuming we read columns with non-numerical characters as string, there would still be the problem that the strings might contain space (or any delimiter) character for some rows. So, each ‘word’ in the string will be interpreted as a column and the program will abort with an error that the rows don’t have the same number of columns.

To correct for these limitations, Gnuastro defines the following convention for storing the table meta-data along with the raw data in one plain text file. The format is primarily designed for ease of reading/writing by eye/fingers, but is also structured enough to be read by a program.

When the first non-white character in a line is #, or there are no non-white characters in it, then the line will not be considered as a row of data in the table (this is a pretty standard convention in many programs, and higher level languages). In the former case, the line is interpreted as a comment. If the comment line starts with ‘# Column N:’, then it is assumed to contain information about column N (a number, counting from 1). Comment lines that don’t start with this pattern are ignored and you can use them to include any further information you want to store with the table in the text file. A column information comment is assumed to have the following format:

```
# Column N: NAME [UNIT, TYPE, BLANK] COMMENT
```

Any sequence of characters between ‘:’ and ‘[’ will be interpreted as the column name (so it can contain anything except the ‘[’ character). Anything between the ‘]’ and the end of the line is defined as a comment. Within the brackets, anything before the first ‘,’ is
the units (physical units, for example km/s, or erg/s), anything before the second ‘,’ is the short type identifier (see below, and Section 4.5 [Numeric data types], page 178). If the type identifier isn’t recognized, the default 64-bit floating point type will be used.

Finally (still within the brackets), any non-white characters after the second ‘,’ are interpreted as the blank value for that column (see Section 6.1.3 [Blank pixels], page 262). The blank value can either be in the same type as the column (for example -99 for a signed integer column), or any string (for example NaN in that same column). In both cases, the values will be stored in memory as Gnuastro’s fixed blank values for each type. For floating point types, Gnuastro’s internal blank value is IEEE NaN (Not-a-Number). For signed integers, it is the smallest possible value and for unsigned integers its the largest possible value.

When a formatting problem occurs, or when the column was already given meta-data in a previous comment, or when the column number is larger than the actual number of columns in the table (the non-commented or empty lines), then the comment information line will be ignored.

When a comment information line can be used, the leading and trailing white space characters will be stripped from all of the elements. For example in this line:

# Column 5: column name [km/s, f32,-99] Redshift as speed

The NAME field will be ‘column name’ and the TYPE field will be ‘f32’. Note how all the white space characters before and after strings are not used, but those in the middle remained. Also, white space characters aren’t mandatory. Hence, in the example above, the BLANK field will be given the value of ‘-99’.

Except for the column number (N), the rest of the fields are optional. Also, the column information comments don’t have to be in order. In other words, the information for column \( N + m \) \((m > 0)\) can be given in a line before column \( N \). Furthermore, you don’t have to specify information for all columns. Those columns that don’t have this information will be interpreted with the default settings (like the case above: values are double precision floating point, and the column has no name, unit, or comment). So these lines are all acceptable for any table (the first one, with nothing but the column number is redundant):

# Column 5:
# Column 1: ID [,i8] The Clump ID.
# Column 3: mag_f160w [AB mag, f32] Magnitude from the F160W filter

The data type of the column should be specified with one of the following values:

- For a numeric column, you can use any of the numeric types (and their recognized identifiers) described in Section 4.5 [Numeric data types], page 178.
- ‘strN’: for strings. The N value identifies the length of the string (how many characters it has). The start of the string on each row is the first non-delimiter character of the column that has the string type. The next N characters will be interpreted as a string and all leading and trailing white space will be removed.

If the next column’s characters, are closer than N characters to the start of the string column in that line/row, they will be considered part of the string column. If there is a new-line character before the ending of the space given to the string column (in other words, the string column is the last column), then reading of the string will stop, even if the N characters are not complete yet. See tests/table/table.txt for one example.
Therefore, the only time you have to pay attention to the positioning and spaces given to the string column is when it is not the last column in the table.

The only limitation in this format is that trailing and leading white space characters will be removed from the columns that are read. In most cases, this is the desired behavior, but if trailing and leading white-spaces are critically important to your analysis, define your own starting and ending characters and remove them after the table has been read. For example in the sample table below, the two ‘|’ characters (which are arbitrary) will remain in the value of the second column and you can remove them manually later.

If only one of the leading or trailing white spaces is important for your work, you can only use one of the ‘|’s.

# Column 1: ID [label, u8]
# Column 2: Notes [no unit, str50]

1  leading and trailing white space is ignored here  2.3442e10
2  |  but they will be preserved here  |  8.2964e11

Note that the FITS binary table standard does not define the unsigned int and unsigned long types, so if you want to convert your tables to FITS binary tables, use other types. Also, note that in the FITS ASCII table, there is only one integer type (long). So if you convert a Gnuastro plain text table to a FITS ASCII table with the Section 5.3 [Table], page 230, program, the type information for integers will be lost. Conversely if integer types are important for you, you have to manually set them when reading a FITS ASCII table (for example with the Table program when reading/converting into a file, or with the gnuastro/table.h library functions when reading into memory).

### 4.7.3 Selecting table columns

At the lowest level, the only defining aspect of a column in a table is its number, or position. But selecting columns purely by number is not very convenient and, especially when the tables are large it can be very frustrating and prone to errors. Hence, table file formats (for example see Section 4.7.1 [Recognized table formats], page 184) have ways to store additional information about the columns (meta-data). Some of the most common pieces of information about each column are its name, the units of data in it, and a comment for longer/informal description of the column’s data.

To facilitate research with Gnuastro, you can select columns by matching, or searching in these three fields, besides the low-level column number. To view the full list of information on the columns in the table, you can use the Table program (see Section 5.3 [Table], page 230) with the command below (replace table-file with the filename of your table, if its FITS, you might also need to specify the HDU/extension which contains the table):

```
$ asttable --information table-file
```

Gnuastro’s programs need the columns for different purposes, for example in Crop, you specify the columns containing the central coordinates of the crop centers with the --coordcol option (see Section 6.1.4.1 [Crop options], page 264). On the other hand, in MakeProfiles, to specify the column containing the profile position angles, you must use the --pcol option (see Section 8.1.4.1 [MakeProfiles catalog], page 448). Thus, there can be no unified common option name to select columns for all programs (different columns have different purposes). However, when the program expects a column for a specific context,
the option names end in the col suffix like the examples above. These options accept values in integer (column number), or string (metadata match/search) format.

If the value can be parsed as a positive integer, it will be seen as the low-level column number. Note that column counting starts from 1, so if you ask for column 0, the respective program will abort with an error. When the value can’t be interpreted as an integer number, it will be seen as a string of characters which will be used to match/search in the table’s meta-data. The meta-data field which the value will be compared with can be selected through the --searchin option, see Section 4.1.2.1 [Input/Output options], page 157. --searchin can take three values: name, unit, comment. The matching will be done following this convention:

- If the value is enclosed in two slashes (for example -x/RA_/ or --coordcol=/RA_/ see Section 6.1.4.1 [Crop options], page 264), then it is assumed to be a regular expression with the same convention as GNU AWK. GNU AWK has a very well written chapter (https://www.gnu.org/software/gawk/manual/html_node/Regexp.html) describing regular expressions, so we will not continue discussing them here. Regular expressions are a very powerful tool in matching text and useful in many contexts. We thus strongly encourage reviewing this chapter for greatly improving the quality of your work in many cases, not just for searching column meta-data in Gnuastro.

- When the string isn’t enclosed between ‘/’s, any column that exactly matches the given value in the given field will be selected.

Note that in both cases, you can ignore the case of alphabetic characters with the --ignorecase option, see Section 4.1.2.1 [Input/Output options], page 157. Also, in both cases, multiple columns may be selected with one call to this function. In this case, the order of the selected columns (with one call) will be the same order as they appear in the table.

4.8 Tessellation

It is sometimes necessary to classify the elements in a dataset (for example pixels in an image) into a grid of individual, non-overlapping tiles. For example when background sky gradients are present in an image, you can define a tile grid over the image. When the tile sizes are set properly, the background’s variation over each tile will be negligible, allowing you to measure (and subtract) it. In other cases (for example spatial domain convolution in Gnuastro, see Section 6.3 [Convolve], page 306), it might simply be for speed of processing: each tile can be processed independently on a separate CPU thread. In the arts and mathematics, this process is formally known as tessellation (https://en.wikipedia.org/wiki/Tessellation).

The size of the regular tiles (in units of data-elements, or pixels in an image) can be defined with the --tilesize option. It takes multiple numbers (separated by a comma) which will be the length along the respective dimension (in FORTRAN/FITS dimension order). Divisions are also acceptable, but must result in an integer. For example --tilesize=30,40 can be used for an image (a 2D dataset). The regular tile size along the first FITS axis (horizontal when viewed in SAO DS9) will be 30 pixels and along the second it will be 40 pixels. Ideally, --tilesize should be selected such that all tiles in the image have exactly
the same size. In other words, that the dataset length in each dimension is divisible by the tile size in that dimension.

However, this is not always possible: the dataset can be any size and every pixel in it is valuable. In such cases, Gnuastro will look at the significance of the remainder length, if it is not significant (for example one or two pixels), then it will just increase the size of the first tile in the respective dimension and allow the rest of the tiles to have the required size. When the remainder is significant (for example one pixel less than the size along that dimension), the remainder will be added to one regular tile’s size and the large tile will be cut in half and put in the two ends of the grid/tessellation. In this way, all the tiles in the central regions of the dataset will have the regular tile sizes and the tiles on the edge will be slightly larger/smaller depending on the remainder significance. The fraction which defines the remainder significance along all dimensions can be set through `--remainderfrac`.

The best tile size is directly related to the spatial properties of the property you want to study (for example, gradient on the image). In practice we assume that the gradient is not present over each tile. So if there is a strong gradient (for example in long wavelength ground based images) or the image is of a crowded area where there isn’t too much blank area, you have to choose a smaller tile size. A larger mesh will give more pixels and so the scatter in the results will be less (better statistics).

For raw image processing, a single tessellation/grid is not sufficient. Raw images are the unprocessed outputs of the camera detectors. Modern detectors usually have multiple readout channels each with its own amplifier. For example the Hubble Space Telescope Advanced Camera for Surveys (ACS) has four amplifiers over its full detector area dividing the square field of view to four smaller squares. Ground based image detectors are not exempt, for example each CCD of Subaru Telescope’s Hyper Suprime-Cam camera (which has 104 CCDs) has four amplifiers, but they have the same height of the CCD and divide the width by four parts.

The bias current on each amplifier is different, and initial bias subtraction is not perfect. So even after subtracting the measured bias current, you can usually still identify the boundaries of different amplifiers by eye. See Figure 11(a) in Akhlaghi and Ichikawa (2015) for an example. This results in the final reduced data to have non-uniform amplifier-shaped regions with higher or lower background flux values. Such systematic biases will then propagate to all subsequent measurements we do on the data (for example photometry and subsequent stellar mass and star formation rate measurements in the case of galaxies).

Therefore an accurate analysis requires a two layer tessellation: the top layer contains larger tiles, each covering one amplifier channel. For clarity we’ll call these larger tiles “channels”. The number of channels along each dimension is defined through the `--numchannels`. Each channel is then covered by its own individual smaller tessellation (with tile sizes determined by the `--tilesize` option). This will allow independent analysis of two adjacent pixels from different channels if necessary. If the image is processed or the detector only has one amplifier, you can set the number of channels in both dimension to 1.

The final tessellation can be inspected on the image with the `--checktiles` option that is available to all programs which use tessellation for localized operations. When this option is called, a FITS file with a `_tiled.fits` suffix will be created along with the outputs, see Section 4.9 [Automatic output], page 191. Each pixel in this image has the number of the tile that covers it. If the number of channels in any dimension are larger than unity, you
will notice that the tile IDs are defined such that the first channels is covered first, then
the second and so on. For the full list of processing-related common options (including
tesselation options), please see Section 4.1.2.2 [Processing options], page 159.

4.9 Automatic output

All the programs in Gnuastro are designed such that specifying an output file or directory
(based on the program context) is optional. When no output name is explicitly given (with
--output, see Section 4.1.2.1 [Input/Output options], page 157), the programs will automati-
cally set an output name based on the input name(s) and what the program does. For exam-
ple when you are using ConvertType to save FITS image named dataset.fits to a JPEG
image and don’t specify a name for it, the JPEG output file will be name dataset.jpg. When
the input is from the standard input (for example a pipe, see Section 4.1.4 [Standard
input], page 167), and --output isn’t given, the output name will be the program’s name
(for example converttype.jpg).

Another very important part of the automatic output generation is that all the directory
information of the input file name is stripped off of it. This feature can be disabled with the
--keepinputdir option, see Section 4.1.2.1 [Input/Output options], page 157. It is the
default because astronomical data are usually very large and organized specially with special
file names. In some cases, the user might not have write permissions in those directories.

Let’s assume that we are working on a report and want to process the FITS images from
two projects (ABC and DEF), which are stored in the sub-directories named ABCproject/
and DEFproject/ of our top data directory (/mnt/data). The following shell commands
show how one image from the former is first converted to a JPEG image through Conver-
type and then the objects from an image in the latter project are detected using NoiseChisel.
The text after the # sign are comments (not typed!).

```
$ pwd # Current location
/home/usrename/research/report
$ ls # List directory contents
ABC01.jpg
$ ls /mnt/data/ABCproject # Archive 1
ABC01.fits ABC02.fits ABC03.fits
$ ls /mnt/data/DEFproject # Archive 2
DEF01.fits DEF02.fits DEF03.fits
$ astconvertt /mnt/data/ABCproject/ABC02.fits --output=jpg # Prog 1
$ ls
ABC01.jpg ABC02.jpg
$ astnoisechisel /mnt/data/DEFproject/DEF01.fits
$ ls
ABC01.jpg ABC02.jpg
4.10 Output FITS files

The output of many of Gnuastro’s programs are (or can be) FITS files. The FITS format
has many useful features for storing scientific datasets (cubes, images and tables) along with

---

18 In fact, even if the data is stored on your own computer, it is advised to only grant write permissions to
the super user or root. This way, you won’t accidentally delete or modify your valuable data!
a robust features for archivability. For more on this standard, please see Section 5.1 [Fits], page 194.

As a community convention described in Section 5.1 [Fits], page 194, the first extension of all FITS files produced by Gnuastro’s programs only contains the meta-data that is intended for the file’s extension(s). For a Gnuastro program, this generic meta-data (that is stored as FITS keyword records) is its configuration when it produced this dataset: file name(s) of input(s) and option names, values and comments. Note that when the configuration is too trivial (only input filename, for example the program Section 5.3 [Table], page 230) no meta-data is written in this extension.

FITS keywords have the following limitations in regards to generic option names and values which are described below:

- If a keyword (option name) is longer than 8 characters, the first word in the record (80 character line) is HIERARCH which is followed by the keyword name.
- Values can be at most 75 characters, but for strings, this changes to 73 (because of the two extra ' characters that are necessary). However, if the value is a file name, containing slash (/) characters to separate directories, Gnuastro will break the value into multiple keywords.
- Keyword names ignore case, therefore they are all in capital letters. Therefore, if you want to use Grep to inspect these keywords, use the -i option, like the example below.

  $ astfits image_detected.fits -h0 | grep -i snquant

The keywords above are classified (separated by an empty line and title) as a group titled “ProgramName configuration”. This meta-data extension, as well as all the other extensions (which contain data), also contain have final group of keywords to keep the basic date and version information of Gnuastro, its dependencies and the pipeline that is using Gnuastro (if its under version control).

**DATE**

The creation time of the FITS file. This date is written directly by CFITSIO and is in UT format.

**COMMIT**

Git’s commit description from the running directory of Gnuastro’s programs. If the running directory is not version controlled or libgit2 isn’t installed (see Section 3.1.2 [Optional dependencies], page 122) then this keyword will not be present. The printed value is equivalent to the output of the following command:

  git describe --dirty --always

If the running directory contains non-committed work, then the stored value will have a ‘-dirty’ suffix. This can be very helpful to let you know that the data is not ready to be shared with collaborators or submitted to a journal. You should only share results that are produced after all your work is committed (safely stored in the version controlled history and thus reproducible).

At first sight, version control appears to be mainly a tool for software developers. However progress in a scientific research is almost identical to progress in software development: first you have a rough idea that starts with handful of easy steps. But as the first results appear to be promising, you will have to extend, or generalize, it to make it more robust and work in all the situations your research covers, not just your first test samples. Slowly you will
find wrong assumptions or bad implementations that need to be fixed (‘bugs’ in software development parlance). Finally, when you submit the research to your collaborators or a journal, many comments and suggestions will come in, and you have to address them.

Software developers have created version control systems precisely for this kind of activity. Each significant moment in the project’s history is called a “commit”, see Section 3.2.2 [Version controlled source], page 132. A snapshot of the project in each “commit” is safely stored away, so you can revert back to it at a later time, or check changes/progress. This way, you can be sure that your work is reproducible and track the progress and history. With version control, experimentation in the project’s analysis is greatly facilitated, since you can easily revert back if a brainstorm test procedure fails.

One important feature of version control is that the research result (FITS image, table, report or paper) can be stamped with the unique commit information that produced it. This information will enable you to exactly reproduce that same result later, even if you have made changes/progress. For one example of a research paper’s reproduction pipeline, please see the reproduction pipeline (https://gitlab.com/makhlaghi/NoiseChisel-paper) of the paper (https://arxiv.org/abs/1505.01664) describing Section 7.2 [NoiseChisel], page 362.

---

**CFITSIO**  The version of CFITSIO used (see Section 3.1.1.2 [CFITSIO], page 120).

**WCSLIB**  The version of WCSLIB used (see Section 3.1.1.3 [WCSLIB], page 121). Note that older versions of WCSLIB do not report the version internally. So this is only available if you are using more recent WCSLIB versions.

**GSL**  The version of GNU Scientific Library that was used, see Section 3.1.1.1 [GNU Scientific Library], page 120.

**GNUASTRO**  The version of Gnuastro used (see Section 1.6 [Version numbering], page 7).

Here is one example of the last few lines of an example output.

```plaintext
/ Versions and date

DATE = '...' / file creation date
COMMIT = 'v0-8-g547f6eb' / Commit description in running dir.
CFITSIO = '3.45 ' / CFITSIO version.
WCSLIB = '5.19 ' / WCSLIB version.
GSL = '2.5 ' / GNU Scientific Library version.
GNUASTRO = '0.7 ' / GNU Astronomy Utilities version.
END
```
5 Data containers

The most low-level and basic property of a dataset is how it is stored. To process, archive and transmit the data, you need a container to store it first. From the start of the computer age, different formats have been defined to store data, optimized for particular applications. One format/container can never be useful for all applications: the storage defines the application and vice-versa. In astronomy, the Flexible Image Transport System (FITS) standard has become the most common format of data storage and transmission. It has many useful features, for example multiple sub-containers (also known as extensions or header data units, HDUs) within one file, or support for tables as well as images. Each HDU can store an independent dataset and its corresponding meta-data. Therefore, Gnuastro has one program (see Section 5.1 [Fits], page 194) specifically designed to manipulate FITS HDUs and the meta-data (header keywords) in each HDU.

Your astronomical research does not just involve data analysis (where the FITS format is very useful). For example you want to demonstrate your raw and processed FITS images or spectra as figures within slides, reports, or papers. The FITS format is not defined for such applications. Thus, Gnuastro also comes with the ConvertType program (see Section 5.2 [ConvertType], page 211) which can be used to convert a FITS image to and from (where possible) other formats like plain text and JPEG (which allow two way conversion), along with EPS and PDF (which can only be created from FITS, not the other way round).

Finally, the FITS format is not just for images, it can also store tables. Binary tables in particular can be very efficient in storing catalogs that have more than a few tens of columns and rows. However, unlike images (where all elements/pixels have one data type), tables contain multiple columns and each column can have different properties: independent data types (see Section 4.5 [Numeric data types], page 178) and meta-data. In practice, each column can be viewed as a separate container that is grouped with others in the table. The only shared property of the columns in a table is thus the number of elements they contain. To allow easy inspection/manipulation of table columns, Gnuastro has the Table program (see Section 5.3 [Table], page 230). It can be used to select certain table columns in a FITS table and see them as a human readable output on the command-line, or to save them into another plain text or FITS table.

5.1 Fits

The “Flexible Image Transport System”, or FITS, is by far the most common data container format in astronomy and in constant use since the 1970s. Archiving (future usage, simplicity) has been one of the primary design principles of this format. In the last few decades it has proved so useful and robust that the Vatican Library has also chosen FITS for its “long-term digital preservation” project.\footnote{https://www.vaticanlibrary.va/home.php?pag=progettodigit}

Although the full name of the standard invokes the idea that it is only for images, it also contains complete and robust features for tables. It started off in the 1970s and was formally published as a standard in 1981, it was adopted by the International Astronomical Union (IAU) in 1982 and an IAU working group to maintain its future was defined in 1988. The FITS 2.0 and 3.0 standards were approved in 2000 and 2008 respectively, and the 4.0 draft has also been released recently, please see the FITS standard document web page (https://}
fits.gsfc.nasa.gov/fits_standard.html) for the full text of all versions. Also see the FITS 3.0 standard paper (https://doi.org/10.1051/0004-6361/201015362) for a nice introduction and history along with the full standard.

Many common image formats, for example a JPEG, only have one image/dataset per file, however one great advantage of the FITS standard is that it allows you to keep multiple datasets (images or tables along with their separate meta-data) in one file. In the FITS standard, each data + metadata is known as an extension, or more formally a header data unit or HDU. The HDUs in a file can be completely independent: you can have multiple images of different dimensions/sizes or tables as separate extensions in one file. However, while the standard doesn’t impose any constraints on the relation between the datasets, it is strongly encouraged to group data that are contextually related with each other in one file. For example an image and the table/catalog of objects and their measured properties in that image. Other examples can be images of one patch of sky in different colors (filters), or one raw telescope image along with its calibration data (tables or images).

As discussed above, the extensions in a FITS file can be completely independent. To keep some information (meta-data) about the group of extensions in the FITS file, the community has adopted the following convention: put no data in the first extension, so it is just meta-data. This extension can thus be used to store Meta-data regarding the whole file (grouping of extensions). Subsequent extensions may contain data along with their own separate meta-data. All of Gnuastro’s programs also follow this convention: the main output dataset(s) are placed in the second (or later) extension(s). The first extension contains no data the program’s configuration (input file name, along with all its option values) are stored as its meta-data, see Section 4.10 [Output FITS files], page 191.

The meta-data contain information about the data, for example which region of the sky an image corresponds to, the units of the data, what telescope, camera, and filter the data were taken with, and filter the data were taken with, its observation date, or the software that produced it and its configuration. Without the meta-data, the raw dataset is practically just a collection of numbers and really hard to understand, or connect with the real world (other datasets). It is thus strongly encouraged to supplement your data (at any level of processing) with as much meta-data about your processing/science as possible.

The meta-data of a FITS file is in ASCII format, which can be easily viewed or edited with a text editor or on the command-line. Each meta-data element (known as a keyword generally) is composed of a name, value, units and comments (the last two are optional). For example below you can see three FITS meta-data keywords for specifying the world coordinate system (WCS, or its location in the sky) of a dataset:

```
LATPOLE = -27.805089 / [deg] Native latitude of celestial pole
RADESYS = 'FK5' / Equatorial coordinate system
EQUINOX = 2000.0 / [yr] Equinox of equatorial coordinates
```

However, there are some limitations which discourage viewing/editing the keywords with text editors. For example there is a fixed length of 80 characters for each keyword (its name, value, units and comments) and there are no new-line characters, so on a text editor all the keywords are seen in one line. Also, the meta-data keywords are immediately followed by the data which are commonly in binary format and will show up as strange looking characters on a text editor, and significantly slowing down the processor.
Gnuastro’s Fits program was designed to allow easy manipulation of FITS extensions and meta-data keywords on the command-line while conforming fully with the FITS standard. For example you can copy or cut (copy and remove) HDUs/extensions from one FITS file to another, or completely delete them. It also has features to delete, add, or edit meta-data keywords within one HDU.

5.1.1 Invoking Fits

Fits can print or manipulate the FITS file HDUs (extensions), meta-data keywords in a given HDU. The executable name is astfits with the following general template

\$ astfits [OPTION...] ASTRdata

One line examples:

## View general information about every extension:
\$ astfits image.fits

## Print the header keywords in the second HDU (counting from 0):
\$ astfits image.fits -h1

## Only print header keywords that contain ‘NAXIS’:
\$ astfits image.fits -h1 | grep NAXIS

## Only print the WCS standard PC matrix elements
\$ astfits image.fits -h1 | grep 'PC._.'

## Copy a HDU from input.fits to out.fits:
\$ astfits input.fits --copy=hdu-name --output=out.fits

## Update the OLDKEY keyword value to 153.034:
\$ astfits --update=OLDKEY,153.034,"Old keyword comment"

## Delete one COMMENT keyword and add a new one:
\$ astfits --delete=COMMENT --comment="Anything you like ;-)."

## Write two new keywords with different values and comments:
\$ astfits --write=MYKEY1,20.00,"An example keyword" --write=MYKEY2,fd

When no action is requested (and only a file name is given), Fits will print a list of information about the extension(s) in the file. This information includes the HDU number, HDU name (EXTNAME keyword), type of data (see Section 4.5 [Numeric data types], page 178, and the number of data elements it contains (size along each dimension for images and table rows and columns). Optionally, a comment column is printed for special situations (like a 2D HEALPix grid that is usually stored as a 1D dataset/table). You can use this to get a general idea of the contents of the FITS file and what HDU to use for further processing, either with the Fits program or any other Gnuastro program.

Here is one example of information about a FITS file with four extensions: the first extension has no data, it is a purely meta-data HDU (commonly used to keep meta-data about the whole file, or grouping of extensions, see Section 5.1 [Fits], page 194). The second extension is an image with name IMAGE and single precision floating point type (float32,
see Section 4.5 [Numeric data types], page 178, it has 4287 pixels along its first (horizontal) axis and 4286 pixels along its second (vertical) axis. The third extension is also an image with name MASK. It is in 2-byte integer format (int16) which is commonly used to keep information about pixels (for example to identify which ones were saturated, or which ones had cosmic rays and so on), note how it has the same size as the IMAGE extension. The third extension is a binary table called CATALOG which has 12371 rows and 5 columns (it probably contains information about the sources in the image).

GNU Astronomy Utilities X.X
Run on Day Month DD HH:MM:SS YYYY
-----
HDU (extension) information: ‘image.fits’.
   Column 1: Index (counting from 0).
   Column 2: Name (‘EXTNAME’ in FITS standard).
   Column 3: Image data type or ‘table’ format (ASCII or binary).
   Column 4: Size of data in HDU.
-----
0   n/a   uint8   0
1   IMAGE   float32   4287x4286
2   MASK   int16   4287x4286
3   CATALOG   table_binary   12371x5

If a specific HDU is identified on the command-line with the --hdu (or -h option) and no operation requested, then the full list of header keywords in that HDU will be printed (as if the --printallkeys was called, see below). It is important to remember that this only occurs when --hdu is given on the command-line. The --hdu value given in a configuration file will only be used when a specific operation on keywords requested. Therefore as described in the paragraphs above, when no explicit call to the --hdu option is made on the command-line and no operation is requested (on the command-line or configuration files), the basic information of each HDU/extension is printed.

The operating mode and input/output options to Fits are similar to the other programs and fully described in Section 4.1.2 [Common options], page 156. The options particular to Fits can be divided into two groups: 1) those related to modifying HDUs or extensions (see Section 5.1.1.1 [HDU information and manipulation], page 197), and 2) those related to viewing/modifying meta-data keywords (see Section 5.1.1.2 [Keyword inspection and manipulation], page 201). These two classes of options cannot be called together in one run: you can either work on the extensions or meta-data keywords in any instance of Fits.

5.1.1.1 HDU information and manipulation

Each FITS file header data unit, or HDU (also known as an extension) is an independent dataset (data + meta-data). Multiple HDUs can be stored in one FITS file, see Section 5.1 [Fits], page 194. The general HDU-related options to the Fits program are listed below as two general classes: the first group below focus on HDU information while the latter focus on manipulating (moving or deleting) the HDUs.

The options below print information about the given HDU on the command-line. Thus they cannot be called together in one command (each has its own independent output).
-n  
--numhdus  
Print the number of extensions/HDUs in the given file. Note that this option  
must be called alone and will only print a single number. It is thus useful in  
scripts, for example when you need to do check the number of extensions in a  
FITS file.

For a complete list of basic meta-data on the extensions in a FITS file, don’t use  
any of the options in this section or in Section 5.1.1.2 [Keyword inspection and  
manipulation], page 201. For more, see Section 5.1.1 [Invoking Fits], page 196.

--hastablehdu  
Print 1 (on standard output) if at least one table HDU (ASCII or binary) exists  
in the FITS file. Otherwise (when no table HDU exists in the file), print 0.

--listtablehdus  
Print the names or numbers (when a name doesn’t exist, counting from zero)  
of HDUs that contain a table (ASCII or Binary) on standard output, one per  
line. Otherwise (when no table HDU exists in the file) nothing will be printed.

--hasimagehdu  
Print 1 (on standard output) if at least one image HDU exists in the FITS file.  
Otherwise (when no image HDU exists in the file), print 0.

In the FITS standard, any array with any dimensions is called an “image”,  
therefore this option includes 1, 3 and 4 dimensional arrays too. However, an  
image HDU with zero dimensions (which is usually the first extension and only  
contains metadata) is not counted here.

--listimagehdus  
Print the names or numbers (when a name doesn’t exist, counting from zero)  
of HDUs that contain an image on standard output, one per line. Otherwise  
(when no image HDU exists in the file) nothing will be printed.

In the FITS standard, any array with any dimensions is called an “image”,  
therefore this option includes 1, 3 and 4 dimensional arrays too. However, an  
image HDU with zero dimensions (which is usually the first extension and only  
contains metadata) is not counted here.

--listallhdus  
Print the names or numbers (when a name doesn’t exist, counting from zero)  
of all HDUs within the input file on the standard output, one per line.

--pixelscale  
Print the HDU’s pixel-scale (change in world coordinate for one pixel along  
each dimension) and pixel area or voxel volume. Without the --quiet option,  
the output of --pixelscale has multiple lines and explanations, thus being  
more human-friendly. It prints the file/HDU name, number of dimensions, and  
the units along with the actual pixel scales. Also, when any of the units are  
in degrees, the pixel scales and area/volume are also printed in units of arc-  
seconds. For 3D datasets, the pixel area (on each 2D slice of the 3D cube) is  
printed as well as the voxel volume.
However, in scripts (that are to be run automatically), this human-friendly format is annoying, so when called with the `--quiet` option, only the pixel-scale value(s) along each dimension is(are) printed in one line. These numbers are followed by the pixel area (in the raw WCS units). For 3D datasets, this will be area on each 2D slice. Finally, for 3D datasets, a final number (the voxel volume) is printed. As a summary, in `--quiet` mode, for 2D datasets three numbers are printed and for 3D datasets, 5 numbers are printed. If the dataset has more than 3 dimensions, only the pixel-scale values are printed (no area or volume will be printed).

`--skycoverage`

Print the rectangular area (or 3D cube) covered by the given image/datacube HDU over the Sky in the WCS units. The covered area is reported in two ways: 1) the center and full width in each dimension, 2) the minimum and maximum sky coordinates in each dimension. This is option is thus useful when you want to get a general feeling of a new image/dataset, or prepare the inputs to query external databases in the region of the image (for example with Section 5.4 [Query], page 249).

If run without the `--quiet` option, the values are given with a human-friendly description. For example here is the output of this option on an image taken near the star Castor:

```
$ astfits castor.fits --skycoverage
Input file: castor.fits (hdu: 1)

Sky coverage by center and (full) width:
Center: 113.9149075 31.93759664
Width: 2.41762045 2.67945253

Sky coverage by range along dimensions:
RA 112.7235592 115.1411797
DEC 30.59262123 33.27207376
```

With the `--quiet` option, the values are more machine-friendly (easy to parse). It has two lines, where the first line contains the center/width values and the second line shows the coordinate ranges in each dimension.

```
$ astfits castor.fits --skycoverage --quiet
113.9149075 31.93759664 2.41762045 2.67945253
112.7235592 115.1411797 30.59262123 33.27207376
```

Note that this is a simple rectangle (cube in 3D) definition, so if the image is rotated in relation to the celestial coordinates a general polygon is necessary to exactly describe the coverage. Hence when there is rotation, the reported area will be larger than the actual area containing data, you can visually see the area with the `--align` option of Section 6.4 [Warp], page 328.

`--datasum`

Calculate and print the given HDU’s "datasum" to stdout. The given HDU is specified with the `--hdu` (or `-h`) option. This number is calculated by parsing all the bytes of the given HDU’s data records (excluding keywords). This
option ignores any possibly existing DATASUM keyword in the HDU. For more on DATASUM in the FITS standard, see Section 5.1.1.2 [Keyword inspection and manipulation], page 201, (under the checksum component of --write).

You can use this option to confirm that the data in two different HDUs (possibly with different keywords) is identical. Its advantage over --write=datasum (which writes the DATASUM keyword into the given HDU) is that it doesn’t require write permissions.

The following options manipulate (move/delete) the HDUs in one FITS file or to another FITS file. These options may be called multiple times in one run. If so, the extensions will be copied from the input FITS file to the output FITS file in the given order (on the command-line and also in configuration files, see Section 4.2.2 [Configuration file precedence], page 170). If the separate classes are called together in one run of Fits, then first --copy is run (on all specified HDUs), followed by --cut (again on all specified HDUs), and then --remove (on all specified HDUs).

The --copy and --cut options need an output FITS file (specified with the --output option). If the output file exists, then the specified HDU will be copied following the last extension of the output file (the existing HDUs in it will be untouched). Thus, after Fits finishes, the copied HDU will be the last HDU of the output file. If no output file name is given, then automatic output will be used to store the HDUs given to this option (see Section 4.9 [Automatic output], page 191).

-C STR
--copy=STR
Copy the specified extension into the output file, see explanations above.

-k STR
--cut=STR
Cut (copy to output, remove from input) the specified extension into the output file, see explanations above.

-R STR
--remove=STR
Remove the specified HDU from the input file.

The first (zero-th) HDU cannot be removed with this option. Consider using --copy or --cut in combination with primaryimghdu to not have an empty zero-th HDU. From CFITSIO: “In the case of deleting the primary array (the first HDU in the file) then [it] will be replaced by a null primary array containing the minimum set of required keywords and no data.”. So in practice, any existing data (array) and meta-data in the first extension will be removed, but the number of extensions in the file won’t change. This is because of the unique position the first FITS extension has in the FITS standard (for example it cannot be used to store tables).

--primaryimghdu
Copy or cut an image HDU to the zero-th HDU/extension a file that doesn’t yet exist. This option is thus irrelevant if the output file already exists or the copied/cut extension is a FITS table. For example with the commands below,
first we make sure that out.fits doesn’t exist, then we copy the first extension of in.fits to the zero-th extension of out.fits.

```
$ rm -f out.fits
$ astfits in.fits --copy=1 --primaryimghdu --output=out.fits
```

If we hadn’t used --primaryimghdu, then the zero-th extension of out.fits would have no data, and its second extension would host the copied image (just like any other output of Gnuastro).

### 5.1.1.2 Keyword inspection and manipulation

The meta-data in each header data unit, or HDU (also known as extension, see Section 5.1 [Fits], page 194) is stored as “keyword”s. Each keyword consists of a name, value, unit, and comments. The Fits program (see Section 5.1 [Fits], page 194) options related to viewing and manipulating keywords in a FITS HDU are described below.

First, let’s review the --keyvalue option which should be called separately from the rest of the options described in this section. Also, unlike the rest of the options in this section, with --keyvalue, you can give more than one input file.

```
-l STR[,STR[,...]]
--keyvalue=STR[,STR[,...]]
```

Only print the value of the requested keyword(s): the STRs. --keyvalue can be called multiple times, and each call can contain multiple comma-separated keywords. If more than one file is given, this option uses the same HDU/extension for all of them (value to --hdu). For example, you can get the number of dimensions of the three FITS files in the running directory, as well as the length along each dimension, with this command:

```
$ astfits *.fits --keyvalue=NAXIS,NAXIS1 --keyvalue=NAXIS2
image-a.fits 2 774 672
image-b.fits 2 774 672
image-c.fits 2 387 336
```

If only one input is given, and the --quiet option is activated, the file name is not printed on the first column, only the values of the requested keywords.

```
$ astfits image-a.fits --keyvalue=NAXIS,NAXIS1 \ 
   --keyvalue=NAXIS2 --quiet
  2 774 672
```

The output is internally stored (and finally printed) as a table (with one column per keyword). Therefore just like the Table program, you can use --colinfoinstdout to print the metadata like the example below (also see Section 5.3.3 [Invoking Table], page 238). The keyword metadata (comments and units) are extracted from the comments and units of the keyword in the input files (first file that has a comment or unit). Hence if the keyword doesn’t have units or comments in any of the input files, they will be empty. For more on Gnuastro’s plain-text metadata format, see Section 4.7.2 [Gnuastro text table format], page 185.

```
$ astfits *.fits --keyvalue=NAXIS,NAXIS1,NAXIS2 \ 
   --colinfoinstdout
# Column 1: FILENAME [name,str10,] Name of input file.
Another advantage of a table output is that you can directly write the table to a file. For example if you add `--output=fileinfo.fits`, the information above will be printed into a FITS table. You can also pipe it into Section 5.3 [Table], page 230, to select files based on certain properties, to sort them based on another property, or any other operation that can be done with Table (including Section 5.3.1 [Column arithmetic], page 230). For example with the command below, you can select all the files that have a size larger than 500 pixels in both dimensions.

```
$ astfits *.fits --keyvalue=NAXIS,NAXIS1,NAXIS2 \
   --colinfoinstdout \
   | asttable --range=NAXIS1,500,inf \
   |   --range=NAXIS2,500,inf -cFILENAME
```

Another advantage of a table output is that you can directly write the table to a file. For example if you add `--output=fileinfo.fits`, the information above will be printed into a FITS table. You can also pipe it into Section 5.3 [Table], page 230, to select files based on certain properties, to sort them based on another property, or any other operation that can be done with Table (including Section 5.3.1 [Column arithmetic], page 230). For example with the command below, you can select all the files that have a size larger than 500 pixels in both dimensions.

```
$ astfits $(find /TOP/DIR/ -name "*.fits") --keyvalue=NAXIS2
```

Note that `--colinfoinstdout` is necessary to use column names when piping to other programs (like asttable above). Also, with the `-cFILENAME` option, we are asking Table to only print the final file names (we don’t need the sizes any more).

The commands with multiple files above used `*.fits`, which is only useful when all your FITS files are in the same directory. However, in many cases, your FITS files will be scattered in multiple sub-directories of a certain top-level directory, or you may only want those with more particular file name patterns. A more powerful way to list the input files to `--keyvalue` is to use the `find` program in Unix-like operating systems. For example, with the command below you can search all the FITS files in all the sub-directories of `/TOP/DIR`.

```
astfits $(find /TOP/DIR/ -name "*.fits") --keyvalue=NAXIS2
```

Below we will discuss the options that can be used to manipulate keywords. To see the full list of keywords in a FITS HDU, you can use the `--printallkeys` option. If any of the keyword modification options below are requested (for example `--update`), the headers of the input file/HDU will be changed first, then printed. Keyword modification is done within the input file. Therefore, if you want to keep the original FITS file or HDU intact, it is easiest to create a copy of the file/HDU first and then run Fits on that (for copying a HDU to another file, see Section 5.1.1.1 [HDU information and manipulation], page 197. In

---

<table>
<thead>
<tr>
<th># Column 2: NAXIS</th>
<th>[ ,u8 ,] number of data axes</th>
</tr>
</thead>
<tbody>
<tr>
<td># Column 3: NAXIS1</td>
<td>[ ,u16 ,] length of data axis 1</td>
</tr>
<tr>
<td># Column 4: NAXIS2</td>
<td>[ ,u16 ,] length of data axis 2</td>
</tr>
<tr>
<td>image-a.fits 2</td>
<td>774 672</td>
</tr>
<tr>
<td>image-b.fits 2</td>
<td>774 672</td>
</tr>
<tr>
<td>image-c.fits 2</td>
<td>387 336</td>
</tr>
</tbody>
</table>
the FITS standard, keywords are always uppercase. So case does not matter in the input or output keyword names you specify.

CHECKSUM automatically updated, when present: the keyword modification options will change the contents of the HDU. Therefore, if a CHECKSUM is present in the HDU, after all the keyword modification options have been complete, Fits will also update CHECKSUM before closing the file.

Most of the options can accept multiple instances in one command. For example you can add multiple keywords to delete by calling --delete multiple times, since repeated keywords are allowed, you can even delete the same keyword multiple times. The action of such options will start from the top most keyword.

The precedence of operations are described below. Note that while the order within each class of actions is preserved, the order of individual actions is not. So irrespective of what order you called --delete and --update. First, all the delete operations are going to take effect then the update operations.

1. --delete
2. --rename
3. --update
4. --write
5. --asis
6. --history
7. --comment
8. --date
9. --printallkeys
10. --verify
11. --copykeys

All possible syntax errors will be reported before the keywords are actually written. FITS errors during any of these actions will be reported, but Fits won’t stop until all the operations are complete. If --quitonerror is called, then Fits will immediately stop upon the first error.

If you want to inspect only a certain set of header keywords, it is easiest to pipe the output of the Fits program to GNU Grep. Grep is a very powerful and advanced tool to search strings which is precisely made for such situations. For example if you only want to check the size of an image FITS HDU, you can run:

    $ astfits input.fits | grep NAXIS

FITS STANDARD KEYWORDS: Some header keywords are necessary for later operations on a FITS file, for example BITPIX or NAXIS, see the FITS standard for their full list. If you modify (for example remove or rename) such keywords, the FITS file extension might not be usable any more. Also be careful for the world coordinate system keywords, if you modify or change their values, any future world coordinate system (like RA and Dec) measurements on the image will also change.
The keyword related options to the Fits program are fully described below.

```
-d STR
--delete=STR
```
Delete one instance of the STR keyword from the FITS header. Multiple instances of --delete can be given (possibly even for the same keyword, when its repeated in the meta-data). All keywords given will be removed from the headers in the same given order. If the keyword doesn’t exist, Fits will give a warning and return with a non-zero value, but will not stop. To stop as soon as an error occurs, run with --quitonerror.

```
-r STR,STR
--rename=STR,STR
```
Rename a keyword to a new value (for example --rename=OLDNAME,NEWNAME. STR contains both the existing and new names, which should be separated by either a comma (,) or a space character. Note that if you use a space character, you have to put the value to this option within double quotation marks (") so the space character is not interpreted as an option separator. Multiple instances of --rename can be given in one command. The keywords will be renamed in the specified order. If the keyword doesn’t exist, Fits will give a warning and return with a non-zero value, but will not stop. To stop as soon as an error occurs, run with --quitonerror.

```
-u STR
--update=STR
```
Update a keyword, its value, its comments and its units in the format described below. If there are multiple instances of the keyword in the header, they will be changed from top to bottom (with multiple --update options).
The format of the values to this option can best be specified with an example:

```
--update=KEYWORD,value,"comments for this keyword",unit
```
If there is a writing error, Fits will give a warning and return with a non-zero value, but will not stop. To stop as soon as an error occurs, run with --quitonerror.
The value can be any numerical or string value. Other than the KEYWORD, all the other values are optional. To leave a given token empty, follow the preceding comma (,) immediately with the next. If any space character is present around the commas, it will be considered part of the respective token. So if more than one token has space characters within it, the safest method to specify a value to this option is to put double quotation marks around each individual token that needs it. Note that without double quotation marks, space characters will be seen as option separators and can lead to undefined behavior.

---

2 Some tricky situations arise with values like ‘87095e5’, if this was intended to be a number it will be kept in the header as 8709500000 and there is no problem. But this can also be a shortened Git commit hash. In the latter case, it should be treated as a string and stored as it is written. Commit hashes are very important in keeping the history of a file during your research and such values might arise without you noticing them in your reproduction pipeline. One solution is to use git describe instead of the short hash alone. A less recommended solution is to add a space after the commit hash and Fits will write the value as ‘87095e5 ’ in the header. If you later compare the strings on the shell, the space character will be ignored by the shell in the latter solution and there will be no problem.
-w STR
--write=STR

Write a keyword to the header. For the possible value input formats, comments and units for the keyword, see the --update option above. The special names (first string) below will cause a special behavior:

/ Write a “title” to the list of keywords. A title consists of one blank line and another which is blank for several spaces and starts with a slash (/). The second string given to this option is the “title” or string printed after the slash. For example with the command below you can add a “title” of ‘My keywords’ after the existing keywords and add the subsequent K1 and K2 keywords under it (note that keyword names are not case sensitive).

$ astfits test.fits -h1 --write=/,"My keywords" \
--write=k1,1.23,"My first keyword" \
--write=k2,4.56,"My second keyword"

$ astfits test.fits -h1
[[[ ... truncated ... ]]]

/ My keywords
K1 = 1.23 / My first keyword
K2 = 4.56 / My second keyword
END

Adding a “title” before each contextually separate group of header keywords greatly helps in readability and visual inspection of the keywords. So generally, when you want to add new FITS keywords, its good practice to also add a title before them.

The reason you need to use / as the keyword name for setting a title is that / is the first non-white character.

The title(s) is(are) written into the FITS with the same order that --write is called. Therefore in one run of the Fits program, you can specify many different titles (with their own keywords under them). For example the command below that builds on the previous example and adds another group of keywords named A1 and A2.

$ astfits test.fits -h1 --write=/,"My keywords" \
--write=k1,1.23,"My first keyword" \
--write=k2,4.56,"My second keyword" \
--write=/,"My second group of keywords" \
--write=a1,7.89,"First keyword" \
--write=a2,0.12,"Second keyword"

checksum When nothing is given afterwards, the header integrity keywords DATASUM and CHECKSUM will be calculated and written/updated. The calculation and writing is done fully by CFITSIO, therefore they comply with the FITS standard 4.0\(^3\) that defines these keywords (its Appendix J).

\(^3\) https://fits.gsfc.nasa.gov/standard40/fits_standard40aa-1e.pdf
If a value is given (e.g., `--write=checksum,MyOwnCheckSum`), then CFITSIO won’t be called to calculate these two keywords and the value (as well as possible comment and unit) will be written just like any other keyword. This is generally not recommended since `CHECKSUM` is a reserved FITS standard keyword. If you want to calculate the checksum with another hashing standard manually and write it into the header, it is recommended to use another keyword name.

In the FITS standard, `CHECKSUM` depends on the HDU’s data and header keywords, it will therefore not be valid if you make any further changes to the header after writing the `CHECKSUM` keyword. This includes any further keyword modification options in the same call to the Fits program. However, `DATASUM` only depends on the data section of the HDU/extension, so it is not changed when you add, remove or update the header keywords. Therefore, it is recommended to write these keywords as the last keywords that are written/modified in the extension. You can use the `--verify` option (described below) to verify the values of these two keywords.

`datasum` Similar to `checksum`, but only write the `DATASUM` keyword (that doesn’t depend on the header keywords, only the data).

`-a STR`  
`--asis=STR`

Write the given `STR` exactly as it is, into the given FITS file header with no modifications. If the contents of `STR` does not conform to the FITS standard for keywords, then it may (most probably: it will) corrupt your file and you may not be able to open it any more. So please be very careful with this option. If you want to define the keyword from scratch, it is best to use the `--write` option (see below) and let CFITSIO worry about complying with the FITS standard.

The best way to use this option is when you want to add a keyword from one FITS file to another, unchanged and untouched. Below is an example of such a case that can be very useful sometimes (on the command-line or in scripts):

```
$ key=$(astfits firstimage.fits | grep KEYWORD)
$ astfits --asis="$key" secondimage.fits
```

In particular note the double quotation signs ("`) around the shell variable (`$key`). This is because FITS keyword strings usually have lots of space characters, if this variable is not quoted, the shell will only give the first word in the full keyword to this option, which will definitely be a non-standard FITS keyword and will make it hard to work on the file afterwards. See the “Quoting” section of the GNU Bash manual for more information if your keyword has the special characters $, ‘, or \
.

You can also use `--asis` to copy multiple keywords from one file to another. But the process will be a little more complicated. So we’ll show the process as the simple shell script below. You can customize it in the first block of variable definitions: 1) set the names of the keywords you want to copy: it can be as
many keys as you want, just put a \`\|\' between them. 

2) The input FITS file (where keywords should be read from) and its HDU. 
3) The output FITS file (where keywords should be written to) and its HDU. 
4) Set a “title” for the newly added keywords in the output (so they are visually separate from the existing keywords in the output).

```bash
#!/bin/bash

# Customizations (input, output and key names).
# NOTE: put a \`\|\' between each keyword name.
keys="KEYa\|KEYb\|KEYc\|KEYd"
ifits=original.fits; ihdu=1
ofits=to_write.fits; ohdu=1
title="Keys from $ifits (hdu $ihdu)"

# Read keywords from input and write in output.
oIFS=$IFS
IFS=''
c="astfits $ofits -h$ohdu --write=/,"$title"
astfits $ifits -h$ihdu \ 
  | grep $keys 
  | (while read line; \ 
      do c="$c --asis=""$line""; \ 
        done; eval $c); \ 
IFS=$oIFS
```

Since its not too long, you can also simply put the variable values of the first block into the second, and write it directly on the command line (if its just for one time).

-H STR

--history STR

Add a HISTORY keyword to the header with the given value. A new HISTORY keyword will be created for every instance of this option. If the string given to this option is longer than 70 characters, it will be separated into multiple keyword cards. If there is an error, Fits will give a warning and return with a non-zero value, but will not stop. To stop as soon as an error occurs, run with --quitonerror.

-c STR

--comment STR

Add a COMMENT keyword to the header with the given value. Similar to the explanation for --history above.

-t

--date

Put the current date and time in the header. If the DATE keyword already exists in the header, it will be updated. If there is a writing error, Fits will give a warning and return with a non-zero value, but will not stop. To stop as soon as an error occurs, run with --quitonerror.
--printallkeys
Print the full meta data (keywords, values, units and comments) in the specified
FITS extension (HDU). If this option is called along with any of the other
keyword editing commands, as described above, all other editing commands
take precedence to this. Therefore, it will print the final keywords after all the
editing has been done.

--printkeynames
Print only the keyword names of the specified FITS extension (HDU), one line
per name. This option must be called alone.

--verify
Verify the DATASUM and CHECKSUM data integrity keywords of the FITS standard.
See the description under the checksum (under --write, above) for more on
these keywords.
This option will print Verified for both keywords if they can be verified. Other-
wise, if they don’t exist in the given HDU/extension, it will print NOT-PRESENT,
and if they cannot be verified it will print INCORRECT. In the latter case (when
the keyword values exist but can’t be verified), the Fits program will also return
with a failure.

By default this function will also print a short description of the DATASUM AND
CHECKSUM keywords. You can suppress this extra information with --quiet
option.

--copykeys=INT:INT/STR,STR[,STR]
Copy the desired set of the input’s keyword records, to the to the output (spec-
fied with the --output and --outhdu for the filename and HDU/extension
respectively). The keywords to copy can be given either as a range (in the
format of INT:INT, inclusive) or a list of keyword names as comma-separated
strings (STR,STR), the list can have any number of keyword names. More details
and examples of the two forms are given below:

Range
The given string to this option must be two integers separated
by a colon (:) The first integer must be positive (counting of the
keyword records starts from 1). The second integer may be negative
(zero is not acceptable) or an integer larger than the first.
A negative second integer means counting from the end. So -1 is
the last copy-able keyword (not including the END keyword).
To see the header keywords of the input with a number before them,
you can pipe the output of the FITS program (when it prints all
the keywords in an extension) into the cat program like below:

$ astfits input.fits -h1 | cat -n

List of names
The given string to this option must be a comma separated list of
keyword names. For example, see the command below:

$ astfits input.fits -h1 --copykeys=KEY1,KEY2 \
Please consider the notes below when copying keywords with names:

- If the number of characters in the name is more than 8, CFITSIO will place a HIERARCH before it. In this case simply give the name and don’t give the HIERARCH (which is a constant and not considered part of the keyword name).

- If your keyword name is composed only of digits, don’t give it as the first name given to --copykeys. Otherwise, it will be confused with the range format above. You can safely give an only-digit keyword name as the second, or third requested keywords.

- If the keyword is repeated more than once in the header, currently only the first instance will be copied. In other words, even if you call --copykeys multiple times with the same keyword name, its first instance will be copied. If you need to copy multiple instances of the same keyword, please get in touch with us at bug-gnuastro@gnu.org.

-outhdu The HDU/extension to write the output keywords of --copykeys.

-Q --quitonerror
Quit if any of the operations above are not successful. By default if an error occurs, Fits will warn the user of the faulty keyword and continue with the rest of actions.

-s STR --datetosec STR
Interpret the value of the given keyword in the FITS date format (most generally: YYYY-MM-DDThh:mm:ss.ddd...) and return the corresponding Unix epoch time (number of seconds that have passed since 00:00:00 Thursday, January 1st, 1970). The Thh:mm:ss.ddd... section (specifying the time of day), and also the .ddd... (specifying the fraction of a second) are optional. The value to this option must be the FITS keyword name that contains the requested date, for example --datetosec=DATE-OBS.

This option can also interpret the older FITS date format (DD/MM/YYThh:mm:ss.ddd...) where only two characters are given to the year. In this case (following the GNU C Library), this option will make the following assumption: values 68 to 99 correspond to the years 1969 to 1999, and values 0 to 68 as the years 2000 to 2068.

This is a very useful option for operations on the FITS date values, for example sorting FITS files by their dates, or finding the time difference between two FITS files. The advantage of working with the Unix epoch time is that you don’t have to worry about calendar details (for example the number of days in different months, or leap years, etc).
--wcscoordsys=STR

Convert the coordinate system of the image’s world coordinate system (WCS) to the given coordinate system (STR) and write it into the file given to --output (or an automatically named file if no --output has been given).

For example with the command below, img-eq.fits will have an identical dataset (pixel values) as image.fits. However, the WCS coordinate system of img-eq.fits will be the equatorial coordinate system in the Julian calendar epoch 2000 (which is the most common epoch used today). Fits will automatically extract the current coordinate system of image.fits and as long as its one of the recognized coordinate systems listed below, it will do the conversion.

$ astfits image.fits --coordsys=eq-j2000 --output=img-eq.fits

The currently recognized coordinate systems are listed below (the most common one today is eq-j2000):

- eq-j2000 2000.0 (Julian-year) equatorial coordinates.
- eq-b1950 1950.0 (Besselian-year) equatorial coordinates.
- ec-j2000 2000.0 (Julian-year) ecliptic coordinates.
- ec-b1950 1950.0 (Besselian-year) ecliptic coordinates.
- galactic Galactic coordinates.
- supergalactic Supergalactic coordinates.

The Equatorial and Ecliptic coordinate systems are defined by the mean equator and equinox epoch: either the Besselian year 1950.0, or the Julian year 2000. For more on their difference and links for further reading about epochs in astronomy, please see the description in Wikipedia (https://en.wikipedia.org/wiki/Epoch_(astronomy)).

--wcsdistortion=STR

If the argument has a WCS distortion, the output (file given with the --output option) will have the distortion given to this option (for example SIP, TPV). The output will be a new file (with a copy of the image, and the new WCS), so if it already exists, the file will be delete (unless you use the --dontdelete option, see Section 4.1.2.1 [Input/Output options], page 157).

With this option, the FITS program will read the minimal set of keywords from the input HDU and the HDU data, it will then write them into the file given to the --output option but with a newly created set of WCS-related keywords corresponding to the desired distortion standard.

If no --output file is specified, an automatically generated output name will be used which is composed of the input’s name but with the -DDD.fits suffix, see Section 4.9 [Automatic output], page 191. Where DDD is the value given to this option (desired output distortion).

Note that all possible conversions between all standards are not yet supported. If the requested conversion is not supported, an informative error message will be printed. If this happens, please let us know and we’ll try our best to add the respective conversions.
For example with the command below, you can be sure that if `in.fits` has a distortion in its WCS, the distortion of `out.fits` will be in the SIP standard.

```bash
$ astfits in.fits --wcsdistortion=SIP --output=out.fits
```

## 5.2 ConvertType

The FITS format used in astronomy was defined mainly for archiving, transmission, and processing. In other situations, the data might be useful in other formats. For example, when you are writing a paper or report, or if you are making slides for a talk, you can’t use a FITS image. Other image formats should be used. In other cases you might want your pixel values in a table format as plain text for input to other programs that don’t recognize FITS. ConvertType is created for such situations. The various types will increase with future updates and based on need.

The conversion is not only one way (from FITS to other formats), but two ways (except the EPS and PDF formats\(^4\)). So you can also convert a JPEG image or text file into a FITS image. Basically, other than EPS/PDF, you can use any of the recognized formats as different color channel inputs to get any of the recognized outputs. So before explaining the options and arguments (in Section 5.2.5 [Invoking ConvertType], page 224), we’ll start with a short description of the recognized files types in Section 5.2.1 [Recognized file formats], page 211, followed a short introduction to digital color in Section 5.2.2 [Color], page 214.

### 5.2.1 Recognized file formats

The various standards and the file name extensions recognized by ConvertType are listed below. Currently Gnuastro uses the file name’s suffix to identify the format.

#### FITS or IMH

Astronomical data are commonly stored in the FITS format (or the older data IRAF .imh format), a list of file name suffixes which indicate that the file is in this format is given in Section 4.1.1.1 [Arguments], page 154.

Each image extension of a FITS file only has one value per pixel/element. Therefore, when used as input, each input FITS image contributes as one color channel. If you want multiple extensions in one FITS file for different color channels, you have to repeat the file name multiple times and use the `--hdu`, `--hdu2`, `--hdu3` or `--hdu4` options to specify the different extensions.

#### JPEG

The JPEG standard was created by the Joint photographic experts group. It is currently one of the most commonly used image formats. Its major advantage is the compression algorithm that is defined by the standard. Like the FITS standard, this is a raster graphics format, which means that it is pixelated. A JPEG file can have 1 (for gray-scale), 3 (for RGB) and 4 (for CMYK) color channels. If you only want to convert one JPEG image into other formats, there is no problem, however, if you want to use it in combination with other input files, make sure that the final number of color channels does not exceed four. If it does, then ConvertType will abort and notify you.

The file name endings that are recognized as a JPEG file for input are: `.jpg`, `.JPEG`, `.jpeg`, `.JEPG`, `.jpe`, `.jif`, `.jfif` and `.jfi`.

---

\(^4\) Because EPS and PDF are vector, not raster/pixelated formats
TIFF

TIFF (or Tagged Image File Format) was originally designed as a common format for scanners in the early 90s and since then it has grown to become very general. In many aspects, the TIFF standard is similar to the FITS image standard: it can allow data of many types (see Section 4.5 [Numeric data types], page 178), and also allows multiple images to be stored in a single file (each image in the file is called a ‘directory’ in the TIFF standard). However, unlike FITS, it can only store images, it has no constructs for tables. Another (inconvenient) difference with the FITS standard is that keyword names are stored as numbers, not human-readable text.

However, outside of astronomy, because of its support of different numeric data types, many fields use TIFF images for accurate (for example 16-bit integer or floating point for example) imaging data.

Currently ConvertType can only read TIFF images, if you are interested in writing TIFF images, please get in touch with us.

EPS

The Encapsulated PostScript (EPS) format is essentially a one page PostScript file which has a specified size. PostScript also includes non-image data, for example lines and texts. It is a fully functional programming language to describe a document. Therefore in ConvertType, EPS is only an output format and cannot be used as input. Contrary to the FITS or JPEG formats, PostScript is not a raster format, but is categorized as vector graphics.

The Portable Document Format (PDF) is currently the most common format for documents. Some believe that PDF has replaced PostScript and that PostScript is now obsolete. This view is wrong, a PostScript file is an actual plain text file that can be edited like any program source with any text editor. To be able to display its programmed content or print, it needs to pass through a processor or compiler. A PDF file can be thought of as the processed output of the compiler on an input PostScript file. PostScript, EPS and PDF were created and are registered by Adobe Systems.

With these features in mind, you can see that when you are compiling a document with \TeX or \LaTeX, using an EPS file is much more low level than a JPEG and thus you have much greater control and therefore quality. Since it also includes vector graphic lines we also use such lines to make a thin border around the image to make its appearance in the document much better. No matter the resolution of the display or printer, these lines will always be clear and not pixelated. In the future, addition of text might be included (for example labels or object IDs) on the EPS output. However, this can be done better with tools within \TeX or \LaTeX such as PGF/Tikz\textsuperscript{5}.

If the final input image (possibly after all operations on the flux explained below) is a binary image or only has two colors of black and white (in segmentation maps for example), then PostScript has another great advantage compared to other formats. It allows for 1 bit pixels (pixels with a value of 0 or 1), this can decrease the output file size by 8 times. So if a gray-scale image is binary, ConvertType will exploit this property in the EPS and PDF (see below) outputs.

\textsuperscript{5} \url{http://sourceforge.net/projects/pgf/}
The standard formats for an EPS file are `.eps`, `.EPS`, `.epsf` and `.epsi`. The EPS outputs of ConvertType have the `.eps` suffix.

**PDF**

As explained above, a PDF document is a static document description format, viewing its result is therefore much faster and more efficient than PostScript. To create a PDF output, ConvertType will make a PostScript page description and convert that to PDF using GPL Ghostscript. The suffixes recognized for a PDF file are: `.pdf`, `.PDF`. If GPL Ghostscript cannot be run on the PostScript file, it will remain and a warning will be printed.

**blank**

This is not actually a file type! But can be used to fill one color channel with a blank value. If this argument is given for any color channel, that channel will not be used in the output.

**Plain text**

Plain text files have the advantage that they can be viewed with any text editor or on the command-line. Most programs also support input as plain text files. As input, each plain text file is considered to contain one color channel.

In ConvertType, the recognized extensions for plain text files are `.txt` and `.dat`. As described in Section 5.2.5 [Invoking ConvertType], page 224, if you just give these extensions, (and not a full filename) as output, then automatic output will be performed to determine the final output name (see Section 4.9 [Automatic output], page 191). Besides the PDF file format cannot be recognized from its name, ConvertType will fall back to plain text mode. So you can use any name (even without an extension) for a plain text input or output. Just note that when the suffix is not recognized, automatic output will not be performed.

The basic input/output on plain text images is very similar to how tables are read/written as described in Section 4.7.2 [Gnuastro text table format], page 185. Simply put, the restrictions are very loose, and there is a convention to define a name, units, data type (see Section 4.5 [Numeric data types], page 178), and comments for the data in a commented line. The only difference is that as a table, a text file can contain many datasets (columns), but as a 2D image, it can only contain one dataset. As a result, only one information comment line is necessary for a 2D image, and instead of the starting '# Column N' (N is the column number), the information line for a 2D image must start with '# Image 1'. When ConvertType is asked to output to plain text file, this information comment line is written before the image pixel values.

When converting an image to plain text, consider the fact that if the image is large, the number of columns in each line will become very large, possibly making it very hard to open in some text editors.

**Standard output (command-line)**

This is very similar to the plain text output, but instead of creating a file to keep the printed values, they are printed on the command line. This can be very useful when you want to redirect the results directly to another program in one command with no intermediate file. The only difference is that only the pixel values are printed (with no information comment line). To print to the standard output, set the output name to `stdout`. 
5.2.2 Color

Color is defined by mixing various measurements/filters. In digital monitors or common digital cameras, colors are displayed/stored by mixing the three basic colors of red, green and blue (RGB) with various proportions. When printing on paper, standard printers use the cyan, magenta, yellow and key (CMYK, key=black) color space. In other words, for each displayed/printed pixel of a color image, the dataset/image has three or four values.

To store/show the three values for each pixel, cameras and monitors allocate a certain fraction of each pixel’s area to red, green and blue filters. These three filters are thus built into the hardware at the pixel level. However, because measurement accuracy is very important in scientific instruments, and we want to do measurements (take images) with various/custom filters (without having to order a new expensive detector!), scientific detectors use the full area of the pixel to store one value for it in a single/mono channel dataset. To make measurements in different filters, we just place a filter in the light path before the detector. Therefore, the FITS format that is used to store astronomical datasets is inherently a mono-channel format (see Section 5.2.1 [Recognized file formats], page 211, or Section 5.1 [Fits], page 194).

When a subject has been imaged in multiple filters, you can feed each different filter into the red, green and blue channels and obtain a colored visualization. In ConvertType, you can do this by giving each separate single-channel dataset (for example in the FITS image format) as an argument (in the proper order), then asking for the output in a format that supports multi-channel datasets (for example JPEG or PDF, see the examples in Section 5.2.5 [Invoking ConvertType], page 224).

As discussed above, color is not defined when a dataset/image contains a single value for each pixel. However, we interact with scientific datasets through monitors or printers (which allow multiple values per pixel and produce color with them). As a result, there is a lot of freedom in visualizing a single-channel dataset. The most basic is to use shades of black (because of its strong contrast with white). This scheme is called grayscale. To help in visualization, more complex mappings can be defined. For example, the values can be scaled to a range of 0 to 360 and used as the “Hue” term of the Hue-Saturation-Value (HSV) color space (while fixing the “Saturation” and “Value” terms). In ConvertType, you can use the --colormap option to choose between different mappings of mono-channel inputs, see Section 5.2.5 [Invoking ConvertType], page 224.

Since grayscale is a commonly used mapping of single-valued datasets, we’ll continue with a closer look at how it is stored. One way to represent a gray-scale image in different color spaces is to use the same proportions of the primary colors in each pixel. This is the common way most FITS image viewers work: for each pixel, they fill all the channels with the single value. While this is necessary for displaying a dataset, there are downsides when storing/saving this type of grayscale visualization (for example in a paper).

- Three (for RGB) or four (for CMYK) values have to be stored for every pixel, this makes the output file very heavy (in terms of bytes).
- If printing, the printing errors of each color channel can make the printed image slightly more blurred than it actually is.
To solve both these problems when storing grayscale visualization, the best way is to save a single-channel dataset into the black channel of the CMYK color space. The JPEG standard is the only common standard that accepts CMYK color space.

The JPEG and EPS standards set two sizes for the number of bits in each channel: 8-bit and 12-bit. The former is by far the most common and is what is used in ConvertType. Therefore, each channel should have values between 0 to $2^8 - 1 = 255$. From this we see how each pixel in a gray-scale image is one byte (8 bits) long, in an RGB image, it is 3 bytes long and in CMYK it is 4 bytes long. But thanks to the JPEG compression algorithms, when all the pixels of one channel have the same value, that channel is compressed to one pixel. Therefore a Grayscale image and a CMYK image that has only the K-channel filled are approximately the same file size.

### 5.2.3 Aligning images with small WCS offsets

In order to have nice color images, it is important that the images be properly aligned. This is usually the case in many scenarios, but it some times happens that the images have a small WCS offset, even though they have the same size. In such cases you can use the script below to align the images into approximately the same pixel grid (to within about 0.5 pixels which is sufficient in many color-image usage scenarios).

The script below does the job using Gnuastro’s Section 6.4 [Warp], page 328, and Section 6.1 [Crop], page 259, programs. Simply copy the lines below into a plain-text file with your favorite text editor and save it as `my-align.sh`. Don’t forget to set the variables of the first three lines to specify the file names (without the `.fits` suffix) and the HDUs of your inputs. These four lines are all you need to edit, leave the rest unchanged. Also, if you are copy/pasting the script from a PDF, be careful that the single-quotes used in AWK may need to be corrected.

```bash
#!/bin/sh

# Set the input names (without the '.fits' suffix),
# and their HDUs.
r=RED_IMAGE_NO_SUFFIX; rhdu=1
g=GREEN_IMAGE_NO_SUFFIX; ghdu=1
b=BLUE_IMAGE_NO_SUFFIX; bhdu=1

# To stop the script if there is a crash
set -e

# Align all the images to the celestial poles.
astwarp $r.fits --align -h$rhdu -o $r-aligned.fits
astwarp $g.fits --align -h$ghdu -o $g-aligned.fits
astwarp $b.fits --align -h$bhdu -o $b-aligned.fits

# Calculate the final WCS-based center and image-based width based on
# the G-band (in RGB) image.
centerwcs=$(astfits $g-aligned.fits --skycoverage --quiet 
    | awk 'NR==1{printf "%g %g", $1,$2}')
widthpix=$(astfits $g-aligned.fits -h1 --quiet 
    | awk 'NR==1{printf "%g %g", $1,$2}')
```

5.2.3 Aligning images with small WCS offsets

In order to have nice color images, it is important that the images be properly aligned. This is usually the case in many scenarios, but it some times happens that the images have a small WCS offset, even though they have the same size. In such cases you can use the script below to align the images into approximately the same pixel grid (to within about 0.5 pixels which is sufficient in many color-image usage scenarios).

The script below does the job using Gnuastro’s Section 6.4 [Warp], page 328, and Section 6.1 [Crop], page 259, programs. Simply copy the lines below into a plain-text file with your favorite text editor and save it as `my-align.sh`. Don’t forget to set the variables of the first three lines to specify the file names (without the `.fits` suffix) and the HDUs of your inputs. These four lines are all you need to edit, leave the rest unchanged. Also, if you are copy/pasting the script from a PDF, be careful that the single-quotes used in AWK may need to be corrected.

```bash
#!/bin/sh

# Set the input names (without the '.fits' suffix),
# and their HDUs.
r=RED_IMAGE_NO_SUFFIX; rhdu=1
g=GREEN_IMAGE_NO_SUFFIX; ghdu=1
b=BLUE_IMAGE_NO_SUFFIX; bhdu=1

# To stop the script if there is a crash
set -e

# Align all the images to the celestial poles.
astwarp $r.fits --align -h$rhdu -o $r-aligned.fits
astwarp $g.fits --align -h$ghdu -o $g-aligned.fits
astwarp $b.fits --align -h$bhdu -o $b-aligned.fits

# Calculate the final WCS-based center and image-based width based on
# the G-band (in RGB) image.
centerwcs=$(astfits $g-aligned.fits --skycoverage --quiet 
    | awk 'NR==1{printf "%g %g", $1,$2}')
widthpix=$(astfits $g-aligned.fits -h1 --quiet 
    | awk 'NR==1{printf "%g %g", $1,$2}')
```
--keyvalue=NAXIS1,NAXIS2 \ 
| awk '{printf "%d,%d", $1, $2}')

# Crop all the images around the desired center and width.
for f in $r $g $b; do
  centerpix=$(echo $centerwcs 
  | asttable -c'arith $1 $2 wcs-to-img' 
      --wcsfile=$f-aligned.fits 
  | awk '{printf "%g,%g", $1, $2}')
  astcrop $f-aligned.fits --mode=img --width=$widthpix 
      --center=$centerpix -o$f-use.fits
  rm $f-aligned.fits
done

Once you have saved the file and come back to your command-line you can run the script like this:

$ chmod +x my-align.sh
$ ./my-align.sh

Of course, feel free to hack it and modify it to fit your datasets, like the rest of Gnuastro, this script is released under GNU GPLv.3 and above, see Section 1.3 [Your rights], page 6.

5.2.4 Annotations for figure in paper

To make a nice figure from your FITS images, it is important to show more than merely the raw image (converted to a printer friendly format like PDF or JPEG). Annotations (or visual metadata) over the raw image greatly help the readers clearly see your argument and put the image/result in a larger context. Examples include:

• Coordinates (Right Ascension and Declination) on the edges of the image, so viewers of your paper or presentation slides can get a physical feeling of the field’s sky coverage.
• Thick line that has a fixed tangential size (for example in kilo parsecs) at the redshift/distance of interest.
• Contours over the image to show radio/X-ray emission, over an optical image for example.
• Text or arrows or etc, over certain parts of the image.

Because of the modular philosophy of Gnuastro, ConvertType is only focused on converting your FITS images to printer friendly formats like JPEG or PDF. But to present your results in a slide or paper, you will often need to annotate the raw JPEG or PDF with some of the features above. The good news is that there are many powerful plotting programs that you can use to add such annotations. As a result, there is no point in making a new one, specific to Gnuastro. In this section, we’ll demonstrate this using the very powerful PGFPLOTS package of \LaTeX\.

\footnote{http://mirrors.ctan.org/graphics/pgf/contrib/pgfplots/doc/pgfplots.pdf}
PGFPlots uses the same \LaTeX\ graphic engine that typesets your paper/slide. Therefore when you build your plots and figures using PGFPlots (and its underlying package PGF/TiKZ\textsuperscript{7}) your plots will blend beautifully within your text: same fonts, same colors, same line properties and etc. Since most papers (and presentation slides\textsuperscript{8}) are made with \LaTeX\, PGFPlots is therefore the best tool for those who use \LaTeX\ to create documents. PGFPlots also doesn’t need any extra dependencies beyond a basic/minimal \TeX-live installation, so it is much more reliable than tools like Matplotlib in Python that have hundreds of fast-evolving dependencies\textsuperscript{9}.

To demonstrate this, we’ll create a surface brightness image of a galaxy in the F160W filter of the ABYSS survey\textsuperscript{10}. In the code-block below, let’s make a “build” directory to keep intermediate files and avoid populating the source. Afterwards, we’ll download the full image and crop out a 20 arcmin wide image around the galaxy with the commands below. You can run these commands in an empty directory.

\begin{verbatim}$ mkdir build
$ wget http://cdsarc.u-strasbg.fr/ftp/J/A+A/621/A133/fits/ah_f160w.fits
$ astcrop ah_f160w.fits --center=53.1616278,-27.7802446 --mode=wcs --width=20/3600 --output=build/crop.fits
\end{verbatim}

To better show the low surface brightness (LSB) outskirts, we’ll warp the image, then convert the pixel units to surface brightness with the commands below. It is very important that the warping is done before the conversion to surface brightness (in units of mag/arcsec\textsuperscript{2}), because the definition of surface brightness is non-linear. For more, see the Surface brightness topic of Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

\begin{verbatim}$ zeropoint=25.94
$ astwarp build/crop.fits --centeroncorner --scale=1/3 \\
    --output=build/scaled.fits
$ pixarea=$(astfits build/scaled.fits --pixelscale --quiet \\
    | awk '{print $1*3600*$2*3600}')
$ astarithmetic build/scaled.fits abs $zeropoint counts-to-mag \\
    pixarea log10 2.5 x + --output=build/sb.fits
\end{verbatim}

We are now ready to convert the surface brightness image into a PDF. To better show the LSB features, we’ll also limit the color range with the --fluxlow and --fluxhigh options: all pixels with a surface brightness brighter than 22 mag/arcsec\textsuperscript{2} will be shown as black, and all pixels with a surface brightness fainter than 30 mag/arcsec\textsuperscript{2} will be white. These thresholds are being defined as variables, because we will also need them below (to pass into PGFPlots). We will also set --borderwidth=0, because the coordinate system

\textsuperscript{7} http://mirrors.ctan.org/graphics/pgf/base/doc/pgfmanual.pdf
\textsuperscript{8} To build slides, \LaTeX\ has packages like Beamer, see http://mirrors.ctan.org/macros/latex/contrib/beamer/doc/beameruserguide.pdf
\textsuperscript{9} See Figure 1 of Alliez et al. 2020 at https://arxiv.org/pdf/1905.11123.pdf
\textsuperscript{10} http://research.iac.es/proyecto/abyss
we will add over the image will effectively be a border for the image (separating it from the background).

```bash
$ sblow=22
$ sbhigh=30
$ astconvertt build/sb.fits --colormap=gray --borderwidth=0 --fluxhigh=$sbhigh --fluxlow=$sblow --output=build/sb.pdf
```

Please open `sb.pdf` and have a look. Also, please open `sb.fits` in DS9 (or any other FITS viewer) and play with the color range. Can the surface brightness limits be changed to better show the LSB structure? If so, you are free to change the limits above.

We now have the printable PDF representation of the image, but as discussed above, its not enough for a paper. We’ll add 1) a thick line showing the size of 20 kpc (kilo parsecs) at the redshift of the central galaxy, 2) coordinates and 3) a color bar, showing the surface brightness level of each grayscale level.

To get the first job done, we first need to know the redshift of the central galaxy. To do this, we can use Gnuastro’s Query program to look into all the objects in NED within this image (only asking for the RA, Dec and redshift columns). We will then use the Match program to find the NED entry that corresponds to our galaxy.

```bash
$ astquery ned --dataset=objdir --overlapwith=build/sb.fits --column=ra,dec,z --output=ned.fits
$ astmatch ned.fits -h1 --coord=53.1616278,-27.7802446 --ccol1=RA,Dec --aperture=1/3600
$ redshift=$(asttable ned_matched.fits -cz)
$ echo $redshift
```

Now that we know the redshift of the central object, we can define the coordinates of the thick line that will show the length of 20 kpc at that redshift. It will be a horizontal line (fixed Declination) across a range of RA. The start of this thick line will be located at the top edge of the image (at the 95-percent of the width and height of the image). With the commands below we'll find the three necessary parameters (one declination and two RAs). Just note that in astronomical images, RA increases to the left/east, which is the reason we are using the minimum and + to find the RA starting point.

```bash
$ scalelineinkpc=20
$ coverage=$(astfits build/sb.fits --skycoverage --quiet | awk 'NR==2')
$ scalelinedec=$(echo $coverage | awk '{print $4-(4-$3)*0.05}')
$ scalelinerastart=$(echo $coverage | awk '{print $1+(2-$1)*0.05}')
$ scalelineraend=$(astcosmiccal --redshift=$redshift --arcsectandist | awk '{start='$scalelinerastart'; width='$scalelineinkpc'/$1/3600; print start+width}')
```

To draw coordinates over the image, we need to feed these values into PGFPlots. But manually entering numbers into the PGFPlots source will be very frustrating and prone to many errors! Fortunately there is an easy way to do this: \LaTeX macros. New macros are defined by this \LaTeX command:

```
\newcommand{\macroname}{value}
```

Anywhere that \LaTeX confronts `\macroname`, it will replace `value` when building the output. We will have one file called `macros.tex` in the build directory and define macros based on
those values. We will use the shell’s `printf` command to write these macro definition lines into the macro file. We just have to use double backslashes in the `printf` command, because backslash is a meaningful character for `printf`, but we want to keep one of them. Also, we put a `\n` at the end of each line, otherwise, all the commands will go into a single line of the macro file. We will also place the random ‘ma’ string at the start of all our \LaTeX macros to help identify the macros for this plot.

```bash
$ macros=build/macros.tex
$ printf '\\newcommand{\maScaleDec}'"\$scalelinedec\n" > $macros
$ printf '\\newcommand{\maScaleRAa}'"\$scalelinerastart\n" >> $macros
$ printf '\\newcommand{\maScaleRAb}'"\$scalelineraend\n" >> $macros
$ printf '\\newcommand{\maScaleKpc}'"\$scalelineinkpc\n" >> $macros
$ printf '\\newcommand{\maCenterZ}'"\$redshift\n" >> $macros
```

Please open the macros file after these commands and have a look to see if they do conform to the expected format above. Another set of macros we will need to feed into PGFPlots is the coordinates of the image corners. Fortunately the `coverage` variable found above is also useful here. We just need to extract each item before feeding it into the macros. To do this, we’ll use AWK and keep each value with the temporary shell variable ‘v’.

```bash
$ v=$(echo $coverage | awk '{print $1}')
$ printf '\\newcommand{\maCropRAMin}'"\$v\n" >> $macros
$ v=$(echo $coverage | awk '{print $2}')
$ printf '\\newcommand{\maCropRAMax}'"\$v\n" >> $macros
$ v=$(echo $coverage | awk '{print $3}')
$ printf '\\newcommand{\maCropDecMin}'"\$v\n" >> $macros
$ v=$(echo $coverage | awk '{print $4}')
$ printf '\\newcommand{\maCropDecMax}'"\$v\n" >> $macros
```

Finally, we also need to pass some other numbers to PGFPlots: 1) the major tick distance (in the coordinate axes that will be printed on the edge of the image). We’ll assume 7 ticks for this image. 2) The minimum and maximum surface brightness values that we gave to ConvertType when making the PDF; PGFPlots will define its color-bar based on these two values.

```bash
$ v=$(echo $coverage | awk '{print ($2-$1)/7}')
$ printf '\\newcommand{\maTickDist}'"\$v\n" >> $macros
$ printf '\\newcommand{\maSBlow}'"\$sblow\n" >> $macros
$ printf '\\newcommand{\maSBhigh}'"\$sbhigh\n" >> $macros
```

All the necessary numbers are now ready. Please copy the contents below into a file called `my-figure.tex`. This is the PGFPlots source for this particular plot. Besides the coordinates and scale-line, we will also add some text over the image and an orange arrow pointing to the central object with its redshift printed over it. The parameters are generally human-readable, so you should be able to get a good feeling of every line. There are also comments which will show up as a different color when you copy this into a plain-text editor.

```latex
\begin{tikzpicture}
%% Define the coordinates and colorbar
\begin{axis}[
```
\begin{axis} [at={(0,0)},
  axis on top, 
  x dir=reverse, 
  scale only axis, 
  width=\linewidth, 
  height=\linewidth, 
  minor tick num=10, 
  xmin=\maCropRAMin, 
  xmax=\maCropRAMax, 
  ymin=\maCropDecMin, 
  ymax=\maCropDecMax, 
  enlargelimits=false, 
  every tick/.style={black}, 
  xtick distance=\maTickDist, 
  ytick distance=\maTickDist, 
  yticklabel style={rotate=90}, 
  ylabel={Declination (degrees)}, 
  xlabel={Right Ascension (degrees)}, 
  ticklabel style={font=\small, 
    /pgf/number format/.cd, precision=4,/tikz/.cd}, 
  x label style={at={(axis description cs:0.5,0.02)}, 
    anchor=north,font=\small}, 
  y label style={at={(axis description cs:0.07,0.5)}, 
    anchor=south,font=\small}, 
  colorbar, 
  colormap name=gray, 
  point meta min=\maSBlow, 
  point meta max=\maSBhigh, 
  colorbar style={ 
    at={(1.01,1)}, 
    ylabel={Surface brightness (mag/arcsec$^2$)}, 
    yticklabel style={ 
      /pgf/number format/.cd, precision=1, /tikz/.cd}, 
    y label style={at={(axis description cs:5.3,0.5)}, 
      anchor=south,font=\small}, 
  }, 
]

% Put the image in the proper positions of the plot.
\addplot graphics[ xmin=\maCropRAMin, xmax=\maCropRAMax, 
  ymin=\maCropDecMin, ymax=\maCropDecMax] 
  {sb.pdf};

% Draw the scale factor.
\addplot[black, line width=5, name=scaleline] coordinates 
  {((\maScaleRAa,\maScaleDec) (\maScaleRAb,\maScaleDec)} 
  node [anchor=north west] {\large \maScaleKpc $\text{kpc}$};
Finally, we need another simple \LaTeX source for the main PDF “report” that will host this figure. This can actually be your paper or slides for example. Here, we’ll suffice to the minimal working example.

\begin{document}

You can write anything here.

\begin{figure}
\input{my-figure.tex}
\caption{A demo image.}
\end{figure}

\end{document}
\end{figure}

\%\% Finish the document.
\end{document}

You are now ready to create the PDF. But \LaTeX{} creates many temporary files, so to avoid populating our top-level directory, we'll copy the two .tex files into the build directory, go there and run \LaTeX{}. Before running it though, we'll first delete all the files that have the name pattern `*-figure0*`, these are “external” files created by TiKZ+PGFPlots, including the actual PDF of the figure.

```
$ cp report.tex my-figure.tex build
$ cd build
$ rm -f *-figure0*
$ pdflatex -shell-escape -halt-on-error report.tex
```

You now have the full “report” in report.pdf. Try adding some extra text on top of the figure, or in the caption and re-running the last four commands. Also try changing the 20kpc scale line length to 50kpc, or try changing the redshift, to see how the length and text of the thick scale-line will automatically change. But the good news is that you also have the raw PDF of the figure that you can use in other places. You can see that file in report-figure0.pdf.

In a larger paper, you can add multiple such figures (with different .tex files that are placed in different figure environments with different captions). Each figure will get a number in the build directory. TiKZ also allows setting a file name for each “external” figure (to avoid such numbers that can be annoying if the image orders are changed). PGFPlots is also highly customizable, you can make a lot of changes and customizations. Both TiKZ\footnote{\url{http://mirrors.ctan.org/graphics/pgf/base/doc/pgfmanual.pdf}} and PGFPlots\footnote{\url{http://mirrors.ctan.org/graphics/pgf/contrib/pgfplots/doc/pgfplots.pdf}} have wonderful manuals, so have a look trough them.

### 5.2.4.1 Full script of annotations on figure

In Section 5.2.4 [Annotations for figure in paper], page 216, we each one of the steps to add annotations over an image were described in detail. So if you have understood the steps, but need to add annotations over an image, repeating those steps individually will be annoying. Therefore in this section, we'll summarize all the steps in a single script that you can simply copy-paste into a text editor, configure, and run.

#### Necessary files:

To run this script, you will need an image to crop your object from (here assuming its called `ah_f160w.fits` with a certain zero point) and two `my-figure.tex` and `report.tex` files that were fully included in Section 5.2.4 [Annotations for figure in paper], page 216. Also, we have brought the redshift as a parameter here. But if the center of your image always points to your main object, you can also include the Query command to automatically find the object's redshift from NED. Alternatively, your image may already be cropped, in this case, you can remove the cropping step and

```
# Parameters.
sblow=22 # Minimum surface brightness.
```

\[11\text{http://mirrors.ctan.org/graphics/pgf/base/doc/pgfmanual.pdf}\]
\[12\text{http://mirrors.ctan.org/graphics/pgf/contrib/pgfplots/doc/pgfplots.pdf}\]
sbhigh=30          # Maximum surface brightness.
bdir=build         # Build directory location on filesystem.
numticks=7         # Number of major ticks in each axis.
redshift=0.619      # Redshift of object of interest.
zeropoint=25.94     # Zeropoint of input image.
scalelineinkpc=20   # Length of scale-line (in kilo parsecs).
input=ah_f160w.fits # Name of input (to crop).

# Stop the script in case of a crash.
set -e

# Build directory
if ! [ -d $bdir ]; then mkdir $bdir; fi

# Crop out the desired region.
crop=$bdir/crop.fits
astcrop $input --center=53.1616278,-27.7802446 --mode=wcs \
   --width=20/3600 --output=$crop

# Warp the image to larger pixels to show surface brightness better.
scaled=$bdir/scaled.fits
astwarp $crop --centeroncorner --scale=1/3 --output=$scaled

# Calculate the pixel area and convert image to Surface brightness.
sb=$bdir/sb.fits
pixarea=$(astfits $scaled --pixelscale --quiet \
   | awk '{print $1*3600*$2*3600}')
astarithmetic $scaled abs $zeropoint counts-to-mag \
   $pixarea log10 2.5 x + --output=$sb

# Convert the Surface brightness image into PDF.
sbpdf=$bdir/sb.pdf
astconvertt $sb --colormap=gray --borderwidth=0 \ 
   --fluxhigh=$sbhigh --fluxlow=$sblow --output=$sbpdf

# Specify the coordinates of the scale line (specifying a certain
# width in kpc). We will put it on the top-right side of the image (5% 
# of the full width of the image away from the edge).
coverage=$(astfits $sb --skycoverage --quiet | awk 'NR==2')
scalelinedec=$(echo $coverage | awk '{print $4-($4-$3)*0.05}
   scalelinerastart=$(echo $coverage | awk '{print $4-($4-$3)*0.05}')
scalelineraend=$(astcosmiccal --redshift=$redshift --arcsectandist \
   | awk '{start='$scalelinerastart'; \n   width='$scalelineinkpc'/$1/3600; \n   print start+width}')

# Write the LaTeX macros to use in plot. Start with the thick line
# showing tangential distance.
macros=$bdir/macros.tex
printf '\newcommand{\maScaleDec}'"{$scalelinedec}\n" > $macros
printf '\newcommand{\maScaleRAa}'"{$scalelinerastart}\n" >> $macros
printf '\newcommand{\maScaleRAb}'"{$scalelineraend}\n" >> $macros
printf '\newcommand{\maScaleKpc}'"{$scalelineinkpc}\n" >> $macros
printf '\newcommand{\maCenterZ}'"{$redshift}\n" >> $macros

# Add image extrema for the coordinates.
v=$(echo $coverage | awk '{print $1}')
printf '\newcommand{\maCropRAMin}'"{$v}\n" >> $macros
v=$(echo $coverage | awk '{print $2}')
printf '\newcommand{\maCropRAMax}'"{$v}\n" >> $macros
v=$(echo $coverage | awk '{print $3}')
printf '\newcommand{\maCropDecMin}'"{$v}\n" >> $macros
v=$(echo $coverage | awk '{print $4}')
printf '\newcommand{\maCropDecMax}'"{$v}\n" >> $macros

# Distance between each tick value.
v=$(echo $coverage | awk '{print ($2-$1)/$numticks}')
printf '\newcommand{\maTickDist}'"{$v}\n" >> $macros
printf '\newcommand{\maSBlow}'"{$sblow}\n" >> $macros
printf '\newcommand{\maSBhigh}'"{$sbhigh}\n" >> $macros

# Copy the LaTeX source into the build directory and go there to run
# it and have all the temporary LaTeX files there.
cp report.tex my-figure.tex $bdir
cd $bdir
rm -f *-figure0*
pdflatex -shell-escape -halt-on-error report.tex

5.2.5 Invoking ConvertType

ConvertType will convert any recognized input file type to any specified output type. The
executable name is astconvertt with the following general template

    $ astconvertt [OPTION...] InputFile [InputFile2] ... [InputFile4]

One line examples:

    ## Convert an image in FITS to PDF:
    $ astconvertt image.fits --output=pdf

    ## Similar to before, but use the Viridis color map:
    $ astconvertt image.fits --colormap=viridis --output=pdf

    ## Convert an image in JPEG to FITS (with multiple extensions
    ## if its color):
    $ astconvertt image.jpg -oimage.fits
## Use three plain text 2D arrays to create an RGB JPEG output:
$ astconvertt f1.txt f2.txt f3.fits -o.jpg

## Use two images and one blank for an RGB EPS output:
$ astconvertt M31_r.fits M31_g.fits blank -oeps

## Directly pass input from output of another program through Standard input (not a file).
$ cat 2darray.txt | astconvertt -oimg.fits

The output’s file format will be interpreted from the value given to the --output option. It can either be given on the command-line or in any of the configuration files (see Section 4.2 [Configuration files], page 169). Note that if the output suffix is not recognized, it will default to plain text format, see Section 5.2.1 [Recognized file formats], page 211.

At most four input files (one for each color channel for formats that allow it) are allowed in ConvertType. The first input dataset can either be a file or come from Standard input (see Section 4.1.4 [Standard input], page 167). The order of multiple input files is important. After reading the input file(s) the number of color channels in all the inputs will be used to define which color space to use for the outputs and how each color channel is interpreted.

Some formats can allow more than one color channel (for example in the JPEG format, see Section 5.2.1 [Recognized file formats], page 211). If there is one input dataset (color channel) the output will be gray-scale, if three input datasets (color channels) are given, they are respectively considered to be the red, green and blue color channels. Finally, if there are four color channels they will be cyan, magenta, yellow and black (CMYK colors).

The value to --output (or -o) can be either a full file name or just the suffix of the desired output format. In the former case, it will used for the output. In the latter case, the name of the output file will be set based on the automatic output guidelines, see Section 4.9 [Automatic output], page 191. Note that the suffix name can optionally start a . (dot), so for example --output=.jpg and --output=jpg are equivalent. See Section 5.2.1 [Recognized file formats], page 211.

Besides the common set of options explained in Section 4.1.2 [Common options], page 156, the options to ConvertType can be classified into input, output and flux related options. The majority of the options are to do with the flux range. Astronomical data usually have a very large dynamic range (difference between maximum and minimum value) and different subjects might be better demonstrated with a limited flux range.

**Input:**

-h STR/INT
--hdu=STR/INT

Input HDU name or counter (counting from 0) for each input FITS file. If the same HDU should be used from all the FITS files, you can use the --globalhdu option described below. In ConvertType, it is possible to call the HDU option multiple times for the different input FITS or TIFF files in the same order that they are called on the command-line. Note that in the TIFF standard, one ‘directory’ (similar to a FITS HDU) may contain multiple color channels (for example when the image is in RGB).
Except for the fact that multiple calls are possible, this option is identical to the common \texttt{--hdu} in Section 4.1.2.1 [Input/Output options], page 157. The number of calls to this option cannot be less than the number of input FITS or TIFF files, but if there are more, the extra HDUs will be ignored, note that they will be read in the order described in Section 4.2.2 [Configuration file precedence], page 170.

Unlike CFITSIO, libtiff (which is used to read TIFF files) only recognizes numbers (counting from zero, similar to CFITSIO) for ‘directory’ identification. Hence the concept of names is not defined for the directories and the values to this option for TIFF files must be numbers.

\textbf{-g STR/INT}  
\textbf{--globalhdu=STR/INT}

Use the value given to this option (a HDU name or a counter, starting from 0) for the HDU identifier of all the input FITS files. This is useful when all the inputs are distributed in different files, but have the same HDU in those files.

\textbf{Output:}

\textbf{-w FLT}  
\textbf{--widthincm=FLT}

The width of the output in centimeters. This is only relevant for those formats that accept such a width (not plain text for example). For most digital purposes, the number of pixels is far more important than the value to this parameter because you can adjust the absolute width (in inches or centimeters) in your document preparation program.

\textbf{-b INT}  
\textbf{--borderwidth=INT}

The width of the border to be put around the EPS and PDF outputs in units of PostScript points. There are 72 or 28.35 PostScript points in an inch or centimeter respectively. In other words, there are roughly 3 PostScript points in every millimeter. If you are planning on adding a border, its significance is highly correlated with the value you give to the \texttt{--widthincm} parameter.

Unfortunately in the document structuring convention of the PostScript language, the “bounding box” has to be in units of PostScript points with no fractions allowed. So the border values only have to be specified in integers. To have a final border that is thinner than one PostScript point in your document, you can ask for a larger width in ConvertType and then scale down the output EPS or PDF file in your document preparation program. For example by setting \texttt{width} in your \texttt{includegraphics} command in \TeX{} or \LaTeX{}. Since it is vector graphics, the changes of size have no effect on the quality of your output quality (pixels don’t get different values).

\textbf{-x}  
\textbf{--hex}

Use Hexadecimal encoding in creating EPS output. By default the ASCII85 encoding is used which provides a much better compression ratio. When converted to PDF (or included in \TeX{} or \LaTeX{} which is finally saved as a PDF file), an efficient binary encoding is used which is far more efficient than both of them. The choice of EPS encoding will thus have no effect on the final PDF.
So if you want to transfer your EPS files (for example if you want to submit your paper to arXiv or journals in PostScript), their storage might become important if you have large images or lots of small ones. By default ASCII85 encoding is used which offers a much better compression ratio (nearly 40 percent) compared to Hexadecimal encoding.

-u INT
--quality=INT

The quality (compression) of the output JPEG file with values from 0 to 100 (inclusive). For other formats the value to this option is ignored. Note that only in gray-scale (when one input color channel is given) will this actually be the exact quality (each pixel will correspond to one input value). If it is in color mode, some degradation will occur. While the JPEG standard does support loss-less graphics, it is not commonly supported.

--colormap=STR[,FLT,...]

The color map to visualize a single channel. The first value given to this option is the name of the color map, which is shown below. Some color maps can be configured. In this case, the configuration parameters are optionally given as numbers following the name of the color map for example see hsv. The table below contains the usable names of the color maps that are currently supported:

- **gray**: Grayscale color map. This color map doesn’t have any parameters. The full dataset range will be scaled to 0 and \(2^8 - 1 = 255\) to be stored in the requested format.
- **hsv**: Hue, Saturation, Value color map. If no values are given after the name (``colormap=hsv``), the dataset will be scaled to 0 and 360 for hue covering the full spectrum of colors. However, you can limit the range of hue (to show only a special color range) by explicitly requesting them after the name (for example ```colormap=hsv,20,240``). The mapping of a single-channel dataset to HSV is done through the Hue and Value elements: Lower dataset elements have lower “value” and lower “hue”. This creates darker colors for fainter parts, while also respecting the range of colors.
- **viridis**: Viridis is the default colormap of the popular Matplotlib module of Python and available in many other visualization tools like PGF-Plots.
- **sls**: The SLS color range, taken from the commonly used SAO DS9 (https://ds9.si.edu). The advantage of this color range is that it starts with black, going into dark blue and finishes with the brighter colors of red and white. So unlike the HSV color range, it includes black and white and brighter colors (like yellow, red) show the larger values.

13 https://en.wikipedia.org/wiki/HSL_and_HSV
**sls-inverse**

The inverse of the SLS color map (see above), where the lowest value corresponds to white and the highest value is black. While SLS is good for visualizing on the monitor, SLS-inverse is good for printing.

**--rgbtohsv**

When there are three input channels and the output is in the FITS format, interpret the three input channels as red, green and blue channels (RGB) and convert them to the hue, saturation, value (HSV) color space.

The currently supported output formats of ConvertType don’t have native support for HSV. Therefore this option is only supported when the output is in FITS format and each of the hue, saturation and value arrays can be saved as one FITS extension in the output for further analysis (for example to select a certain color).

Flux range:

-c STR
--change=STR

(=STR) Change pixel values with the following format "from1:to1, from2:to2,...". This option is very useful in displaying labeled pixels (not actual data images which have noise) like segmentation maps. In labeled images, usually a group of pixels have a fixed integer value. With this option, you can manipulate the labels before the image is displayed to get a better output for print or to emphasize on a particular set of labels and ignore the rest. The labels in the images will be changed in the same order given. By default first the pixel values will be converted then the pixel values will be truncated (see --fluxlow and --fluxhigh).

You can use any number for the values irrespective of your final output, your given values are stored and used in the double precision floating point format. So for example if your input image has labels from 1 to 20000 and you only want to display those with labels 957 and 11342 then you can run ConvertType with these options:

```
$ astconvertt --change=957:50000,11342:50001 --fluxlow=5e4 --fluxhigh=1e5 segmentationmap.fits --output=jpg
```

While the output JPEG format is only 8 bit, this operation is done in an intermediate step which is stored in double precision floating point. The pixel values are converted to 8-bit after all operations on the input fluxes have been complete. By placing the value in double quotes you can use as many spaces as you like for better readability.

-C
--changeaftertrunc

Change pixel values (with --change) after truncation of the flux values, by default it is the opposite.
-L FLT
--fluxlow=FLT
The minimum flux (pixel value) to display in the output image, any pixel value below this value will be set to this value in the output. If the value to this option is the same as --fluxhigh, then no flux truncation will be applied. Note that when multiple channels are given, this value is used for all the color channels.

-H FLT
--fluxhigh=FLT
The maximum flux (pixel value) to display in the output image, see --fluxlow.

-m INT
--maxbyte=INT
This is only used for the JPEG and EPS output formats which have an 8-bit space for each channel of each pixel. The maximum value in each pixel can therefore be $2^8 - 1 = 255$. With this option you can change (decrease) the maximum value. By doing so you will decrease the dynamic range. It can be useful if you plan to use those values for other purposes.

-A
--forcemin
Enforce the value of --fluxlow (when its given), even if its smaller than the minimum of the dataset and the output is format supporting color. This is particularly useful when you are converting a number of images to a common image format like JPEG or PDF with a single command and want them all to have the same range of colors, independent of the contents of the dataset. Note that if the minimum value is smaller than --fluxlow, then this option is redundant.

By default, when the dataset only has two values, and the output format is PDF or EPS, ConvertType will use the PostScript optimization that allows setting the pixel values per bit, not byte (Section 5.2.1 [Recognized file formats], page 211). This can greatly help reduce the file size. However, when --fluxlow or --fluxhigh are called, this optimization is disabled: even though there are only two values (is binary), the difference between them does not correspond to the full contrast of black and white.

-B
--forcemax
Similar to --forcemin, but for the maximum.

-i
--invert
For 8-bit output types (JPEG, EPS, and PDF for example) the final value that is stored is inverted so white becomes black and vice versa. The reason for this is that astronomical images usually have a very large area of blank sky in them. The result will be that a large are of the image will be black. Note that this behavior is ideal for gray-scale images, if you want a color image, the colors are going to be mixed up.
5.3 Table

Tables are the high-level products of processing on low-leveler data like images or spectra. For example in Gnuastro, MakeCatalog will process the pixels over an object and produce a catalog (or table) with the properties of each object like magnitudes, positions and etc (see Section 7.4 [MakeCatalog], page 392). Each one of these properties is a column in its output catalog (or table) and for each input object, we have a row.

When there are only a small number of objects (rows) and not too many properties (columns), then a simple plain text file is mainly enough to store, transfer, or even use the produced data. However, to be more efficient, astronomers have defined the FITS binary table standard to store data in a binary format (which cannot be seen in a text editor text). This can offer major advantages: the file size will be greatly reduced and the reading and writing will also be faster (because the RAM and CPU also work in binary). The acceptable table formats are fully described in Section 4.7 [Tables], page 183.

Binary tables are not easily readable with basic plain-text editors. There is no fixed/unified standard on how the zero and ones should be interpreted. Unix-like operating systems have flourished because of a simple fact: communication between the various tools is based on human readable characters. So while the FITS table standards are very beneficial for the tools that recognize them, they are hard to use in the vast majority of available software. This creates limitations for their generic use.

Table is Gnuastro’s solution to this problem. Table has a large set of operations that you can directly do on any recognized table (like selecting certain rows, doing arithmetic on the columns and etc). For operations that Table doesn’t do internally, FITS tables (ASCII or binary) are directly accessible to the users of Unix-like operating systems (in particular those working the command-line or shell, see Section 1.7.1 [Command-line interface], page 9). With Table, a FITS table (in binary or ASCII formats) is only one command away from AWK (or any other tool you want to use). Just like a plain text file that you read with the cat command. You can pipe the output of Table into any other tool for higher-level processing, see the examples in Section 5.3.3 [Invoking Table], page 238, for some simple examples.

In the sections below we describe how to effectively use the Table program. We start with Section 5.3.1 [Column arithmetic], page 230, where the basic concept and methods of applying arithmetic operations on one or more columns are discussed. Afterwards, in Section 5.3.2 [Operation precedence in Table], page 235, we review the various types of operations available and their precedence in an instance of calling Table. This is a good place to get a general feeling of all the things you can do with Table. Finally, in Section 5.3.3 [Invoking Table], page 238, we give some examples and describe each option in Table.

5.3.1 Column arithmetic

In many scenarios, you want to apply some kind of operation on the columns and save them in another table or feed them into another program. With Table you can do a rich set of operations on the contents of one or more columns in a table, and save the resulting values...
as new column(s) in the output table. For seeing the precedence of Column arithmetic in relation to other Table operators, see Section 5.3.2 [Operation precedence in Table], page 235.

To enable column arithmetic, the first 6 characters of the value to --column (-c) should be the activation word ‘arith’ (note the space character in the end, after ‘arith’). After the activation word, you can use reverse polish notation to identify the operators and their operands, see Section 6.2.1 [Reverse polish notation], page 272. Just note that white-space characters are used between the tokens of the arithmetic expression and that they are meaningful to the command-line environment. Therefore the whole expression (including the activation word) has to be quoted on the command-line or in a shell script (see the examples below).

To identify a column you can directly use its name, or specify its number (counting from one, see Section 4.7.3 [Selecting table columns], page 188). When you are giving a column number, it is necessary to prefix the number with a $, similar to AWK. Otherwise the number is not distinguishable from a constant number to use in the arithmetic operation.

For example with the command below, the first two columns of table.fits will be printed along with a third column that is the result of multiplying the first column with 10\(^{10}\) (for example to convert wavelength from Meters to Angstroms). Note that without the $, it is not possible to distinguish between “1” as a column-counter, or “1” as a constant number to use in the arithmetic operation. Also note that because of the significance of $ for the command-line environment, the single-quotes are the recommended quoting method (as in an AWK expression), not double-quotes (for the significance of using single quotes see the box below).

$ asttable table.fits -c1,2 -c'arith $1 1e10 x'

Single quotes when string contains $: On the command-line, or in shell-scripts, $ is used to expand variables, for example echo $PATH prints the value (a string of characters) in the variable PATH, it will not simply print $PATH. This operation is also permitted within double quotes, so echo "$PATH" will produce the same output. This is good when printing values, for example in the command below, $PATH will expand to the value within it.

$ echo "My path is: $PATH"

If you actually want to return the literal string $PATH, not the value in the PATH variable (like the scenario here in column arithmetic), you should put it in single quotes like below. The printed value here will include the $, please try it to see for your self and compare to above.

$ echo 'My path is: $PATH'

Therefore, when your column arithmetic involves the $ sign (to specify columns by number), quote your arith string with a single quotation mark. Otherwise you can use both single or double quotes.

Alternatively, if the columns have meta-data and the first two are respectively called AWAV and SPECTRUM, the command above is equivalent to the command below. Note that the character ‘$’ is no longer necessary in this scenario (because names will not be confused with numbers):

$ asttable table.fits -cAWAV,SPECTRUM -c'arith AWAV 1e10 x'
Comparison of the two commands above clearly shows why it is recommended to use column names instead of numbers. When the columns have descriptive names, the command/script actually becomes much more readable, describing the intent of the operation. It is also independent of the low-level table structure: for the second command, the column numbers of the \texttt{AWAV} and \texttt{SPECTRUM} columns in \texttt{table.fits} is irrelevant.

Column arithmetic changes the values of the data within the column. So the old column meta data can’t be used any more. By default the output column of the arithmetic operation will be given a generic metadata (for example its name will be \texttt{ARITH_1}, which is hardly useful!). But meta data are critically important and it is good practice to always have short, but descriptive, names for each columns, units and also some comments for more explanation. To add metadata to a column, you can use the \texttt{--colmetadata} option that is described in Section 5.3.3 [Invoking Table], page 238, and Section 5.3.2 [Operation precedence in Table], page 235.

Finally, since the arithmetic expressions are a value to \texttt{--column}, it doesn’t necessarily have to be a separate option, so the commands above are also identical to the command below (note that this only has one \texttt{-c} option). Just be very careful with the quoting!

$$\texttt{asttable table.fits -cAWAV,SPECTRUM,\textquote{arith AWAV 1e10 x}}$$

Almost all the arithmetic operators of Section 6.2.3 [Arithmetic operators], page 276, are also supported for column arithmetic in Table. In particular, the few that are not present in the Gnuastro library aren’t yet supported for column arithmetic. Besides the operators in Section 6.2.3 [Arithmetic operators], page 276, several operators are only available in Table to use on table columns.

\texttt{wcs-to-img}

Convert the given WCS positions to image/dataset coordinates based on the number of dimensions in the WCS structure of \texttt{--wcshdu} extension/HDU in \texttt{--wcsfile}. It will output the same number of columns. The first popped operand is the last FITS dimension.

For example the two commands below (which have the same output) will produce 5 columns. The first three columns are the input table’s ID, RA and Dec columns. The fourth and fifth columns will be the pixel positions in \texttt{image.fits} that correspond to each RA and Dec.

$$\texttt{asttable table.fits -cID,RA,DEC,\textquote{arith RA DEC wcs-to-img}' \ --wcsfile=image.fits}$$

$$\texttt{asttable table.fits -cID,RA -cDEC \ -c\textquote{arith RA DEC wcs-to-img}' --wcsfile=image.fits}$$

\texttt{img-to-wcs}

Similar to \texttt{wcs-to-img}, except that image/dataset coordinates are converted to WCS coordinates.

\texttt{distance-flat}

Return the distance between two points assuming they are on a flat surface. Note that each point needs two coordinates, so this operator needs four operands

\footnote{For a list of the Gnuastro library arithmetic operators, please see the macros starting with \texttt{GAL\_ARITHMETIC\_OP} and ending with the operator name in Section 11.3.14 [Arithmetic on datasets (arithmetic.h)], page 600.}
(currently it only works for 2D spaces). The first and second popped operands are considered to belong to one point and the third and fourth popped operands to the second point.

Each of the input points can be a single coordinate or a full table column (containing many points). In other words, the following commands are all valid:

```bash
$ asttable table.fits \\
-`arith X1 Y1 X2 Y2 distance-flat'
$ asttable table.fits \\
-`arith X Y 12.345 6.789 distance-flat'
$ asttable table.fits \\
-`arith 12.345 6.789 X Y distance-flat'
```

In the first case we are assuming that `table.fits` has the following four columns `X1`, `Y1`, `X2`, `Y2`. The returned column by this operator will be the difference between two points in each row with coordinates like the following (\(X1, Y1\)) and (\(X2, Y2\)). In other words, for each row, the distance between different points is calculated. In the second and third cases (which are identical), it is assumed that `table.fits` has the two columns `X` and `Y`. The returned column by this operator will be the difference of each row with the fixed point at (12.345, 6.789).

**distance-on-sphere**

Return the spherical angular distance (along a great circle, in degrees) between the given two points. Note that each point needs two coordinates (in degrees), so this operator needs four operands. The first and second popped operands are considered to belong to one point and the third and fourth popped operands to the second point.

Each of the input points can be a single coordinate or a full table column (containing many points). In other words, the following commands are all valid:

```bash
$ asttable table.fits \\
-`arith RA1 DEC1 RA2 DEC2 distance-on-sphere'
$ asttable table.fits \\
-`arith RA DEC 9.876 5.432 distance-on-sphere'
$ asttable table.fits \\
-`arith 9.876 5.432 RA DEC distance-on-sphere'
```

In the first case we are assuming that `table.fits` has the following four columns `RA1`, `DEC1`, `RA2`, `DEC2`. The returned column by this operator will be the difference between two points in each row with coordinates like the following (\(RA1, DEC1\)) and (\(RA2, DEC2\)). In other words, for each row, the angular distance between different points is calculated. In the second and third cases (which are identical), it is assumed that `table.fits` has the two columns `RA` and `DEC`. The returned column by this operator will be the difference of each row with the fixed point at (9.876, 5.432).
The distance (along a great circle) on a sphere between two points is calculated with the equation below, where $r_1$, $r_2$, $d_1$ and $d_2$ are the right ascensions and declinations of points 1 and 2.

$$\cos(d) = \sin(d_1) \sin(d_2) + \cos(d_1) \cos(d_2) \cos(r_1 - r_2)$$

**ra-to-degree**

Convert the hour-wise Right Ascension (RA) string, in the sexagesimal format of \_h\_m\_s or \_:\_:\_, to degrees. Note that the input column has to have a string format. In FITS tables, string columns are well-defined. For plain-text tables, please follow the standards defined in Section 4.7.2 [Gnuastro text table format], page 185, otherwise the string column won’t be read.

```bash
$ asttable catalog.fits -c'arith RA ra-to-degree'
```

**dec-to-degree**

Convert the sexagesimal Declination (Dec) string, in the format of \_d\_m\_s or \_:\_:\_, to degrees (a single floating point number). For more details please see the ra-to-degree operator.

**degree-to-ra**

Convert degrees (a column with a single floating point number) to the Right Ascension, RA, string (in the sexagesimal format hours, minutes and seconds, written as \_h\_m\_s). The output will be a string column so no further mathematical operations can be done on it. The output file can be in any format (for example FITS or plain-text). If it is plain-text, the string column will be written following the standards described in Section 4.7.2 [Gnuastro text table format], page 185.

**degree-to-dec**

Convert degrees (a column with a single floating point number) to the Declination, Dec, string (in the format of \_d\_m\_s). See the degree-to-ra for more on the format of the output.

**date-to-sec**

Return the number of seconds from the Unix epoch time (00:00:00 Thursday, January 1st, 1970). The input (popped) operand should be a string column in the FITS date format (most generally: YYYY-MM-DDThh:mm:ss.ddd...).

The returned operand will be named UNIXSEC (short for Unix-seconds) and will be a 64-bit, signed integer, see Section 4.5 [Numeric data types], page 178. If the input string has sub-second precision, it will be ignored because floating point numbers cannot accurately store numbers with many significant digits. To preserve sub-second precision, please use date-to-millisec.

For example, in the example below we are using this operator, in combination with the --keyvalue option of the Fits program, to sort your desired FITS files by observation date (value in the DATE-OBS keyword in example below):

```bash
$ astfits *.fits --keyvalue=DATE-OBS --colinfo stdout
```
If you don’t need to see the Unix-seconds any more, you can add a \texttt{--colinfoinstdout} (short for \texttt{--column=FILENAME}) at the end. For more on \texttt{--keyvalue}, see Section 5.1.1.2 [Keyword inspection and manipulation], page 201.

\textbf{date-to-millisecond}

Return the number of milli-seconds from the Unix epoch time (00:00:00 Thursday, January 1st, 1970). The input (popped) operand should be a string column in the FITS date format (most generally: YYYY-MM-DDThh:mm:ss.ddd..., where .ddd is the optional sub-second component).

The returned operand will be named \texttt{UNIXMILLISEC} (short for Unix milliseconds) and will be a 64-bit, signed integer, see Section 4.5 [Numeric data types], page 178. The returned value is not a floating point type because for large numbers, floating point data types loose single-digit precision (which is important here).

Other than the units of the output, this operator behaves similarly to \texttt{date-to-sec}. See the description of that operator for an example.

\section{Operation precedence in Table}

The Table program can do many operations on the rows and columns of the input tables and they aren’t always applied in the order you call the operation on the command-line. In this section we will describe which operation is done before/after which operation. Knowing this precedence table is important to avoid confusion when you ask for more than one operation. For a description of each option, please see Section 5.3.3 [Invoking Table], page 238.

\textbf{Column information (\texttt{--information} or \texttt{-i})}

When given this option, the column data are not read at all. Table simply reads the column metadata (name, units, numeric data type and comments), and the number of rows and prints them. Table then terminates and no other operation is done. This can therefore be called at the end of an arbitrarily long Table command only to remember the column metadata, then deleted to continue writing the command (using the shell’s history to retrieve the previous command with an up-arrow key).

\textbf{Column selection (\texttt{--column})}

When this option is given, only the columns given to this option (from the main input) will be used for all future steps. When \texttt{--column} (or \texttt{-c}) isn’t given, then all the main input’s columns will be used in the next steps.

\textbf{Column(s) from other file(s) (\texttt{--catcolumnfile} and \texttt{--catcolumnhdu}, \texttt{--catcolumns})}

When column concatenation (addition) is requested, columns from other tables (in other files, or other HDUs of the same FITS file) will be added after the existing columns read from the main input. In one command, you can call these options multiple times to allow addition of columns from many files.

The rest of the operations below are done on the rows, therefore you can merge the columns of various tables into one table, then start adding/limiting the rows.
of the output. If any of the row-based operations below are requested in the
same `asttable` command, they will also be applied to the rows of the added
columns. However, the conditions to keep/reject rows can only be applied to
the rows of the columns in main input table (not the columns that are added
with these options).

Rows from other file(s) (`--catrowfile` and `--catrowhdu`)
With this feature, you can import rows from other tables (in other files, or
other HDUs of the same FITS file). The same column selection of `--column`
is applied to the tables given here. The column metadata (name, units and
comments) will be taken from the main input. Two conditions are mandatory
for adding rows:

- The number of columns used from the new tables must be equal to the
  number of columns in memory, by the time control reaches here.
- The data type of each column (see Section 4.5 [Numeric data types],
  page 178) should be the same as the respective column in memory by
  the time control reaches here. If the data types are different, you can use
  the type conversion operators of Table’s column arithmetic on the inputs in
  a separate command first (see Section 6.2.3.12 [Numerical type conversion
  operators], page 296, and Section 5.3.1 [Column arithmetic], page 230).

Row selection by value in a column

- `--range`: only keep rows within a certain interval in given column.
- `--insidepolygon`: only keep rows within the polygon of `--polygon`.
- `--outsidepolygon`: only keep rows outside the polygon of `--polygon`.
- `--equal`: only keep rows with specified value in given column.
- `--notequal`: only keep rows without specified value in given column.
- `--noblank`: only keep rows that aren’t blank in the given column(s).

These options take certain column(s) as input and remove some rows from the
full table (all columns), based on the given limitations. They can be called any
number of times (to limit the final rows based on values in different columns
for example). Since these are row-rejection operations, their internal order is
irrelevant. In other words, it makes no difference if `--equal` is called before or
after `--range` for example.

As a side-effect, because NaN/blank values are defined to fail on any condition,
these operations will also remove rows with NaN/blank values in the specified
column they are checking. Also, the columns that are used for these operations
don’t necessarily have to be in the final output table (you may not need the
column after doing the selection based on it).

Even though these options are applied after merging columns from other tables,
currently their condition-columns can only come from the main input table. In
other words, even though the rows of the added columns (from another file) will
also be selected with these options, the condition to keep/reject rows cannot be
taken from the newly added columns.

These options are applied first because the speed of later operations can be
greatly affected by the number of rows. For example, if you also call the `--sort`
option, and your row selection will result in 50 rows (from an input of 1000 rows), limiting the number of rows can greatly speed up the sorting in your final output.

Sorting (`--sort`)
Sort of the rows based on values in a certain column. The column to sort by can only come from the main input table columns (not columns that may have been added with `--catcolumnfile`).

Row selection (by position)
- `--head`: keep only requested number of top rows.
- `--tail`: keep only requested number of bottom rows.
- `--rowrandom`: keep only a random number of rows.
- `--rowlimit`: keep only rows within a certain positional interval.

These options limit/select rows based on their position within the table (not their value in any certain column).

Column arithmetic
Once the final rows are selected in the requested order, column arithmetic is done (if requested). For more on column arithmetic, see Section 5.3.1 [Column arithmetic], page 230.

Column metadata (`--colmetadata`)
Changing column metadata is necessary after column arithmetic or adding new columns from other tables (that were done above).

Output row selection (`--noblankend`)
Only keep the output rows that don’t have a blank value in the given column(s). For example, you may need to apply arithmetic operations on the columns (through Section 5.3.1 [Column arithmetic], page 230) before rejecting the undesired rows. After the arithmetic operation is done, you can use the `where` operator to set the non-desired columns to NaN/blank and use `--noblankend` option to remove them just before writing the output. In other scenarios, you may want to remove blank values based on columns in another table. You can also use the modified metadata of the previous steps to use updated names! See the example below for applying any generic value-based row selection based on `--noblankend`.

As an example, let’s review how Table interprets the command below. We are assuming that `table.fits` contains at least three columns: RA, DEC and PARAM and you only want the RA and Dec of the rows where $p \times 2 < 5$ ($p$ is the value of each row in the PARAM column).

```
asttable table.fits -cRA,DEC --noblankend=MULTIP \
    -c'arith PARAM 2 x set-i i i 5 gt nan where' \
    --colmetadata=3,MULTIP,unit,"Description of column"
```

Due to the precedence described in this section, Table does these operations (which are independent of the order of the operations written on the command-line):

1. At the start (with `-cRA,DEC`), Table reads the RA and DEC columns.
2. In between all the operations in the command above, Column arithmetic (with `c’arith ...‘) has the highest precedence. So the arithmetic operation is done and stored as a new (third) column. In this arithmetic operation, we multiply all the values of the PARAM column by 2, then set all those with a value larger than 5 to NaN (for more on understanding this operation, see the ‘set-’ and ‘where’ operators in Section 6.2.3 [Arithmetic operators], page 276).

3. Updating column metadata (with --colmetadata) is then done to give a name (MULTIP) to the newly calculated (third) column. During the process, besides a name, we also set a unit and description for the new column. These metadata entries are very important, so always be sure to add metadata after doing column arithmetic.

4. The lowest precedence operation is --noblankend=MULTIP. So only rows that aren’t blank/NaN in the MULTIP column are kept.

5. Finally, the output table (with three columns) is written to the command-line. If you also want to print the column metadata, you can use the --colinfoinstdout option. Alternatively, if you want the output in a file, you can use the --output option to save the table in FITS or plain-text format.

Out of precedence: It may happen that your desired operation needs a separate precedence. In this case you can pipe the output of Table into another call of Table and use the --colinfoinstdout option to preserve the metadata between the two calls.

For example, let’s assume that you want to sort the output table from the example command above based on the new MULTIP column. Since sorting is done prior to column arithmetic, you can’t do it in one command, but you can circumvent this limitation by simply piping the output (including metadata) to another call to Table:

```
asttable table.fits -cRA,DEC --noblankend=MULTIP --colinfoinstdout \
  -c'arith PARAM 2 x set-i i i 5 gt nan where' \
  --colmetadata=3,MULTIP,unit,"Description of column" \
| asttable --sort=MULTIP --output=selected.fits
```

5.3.3 Invoking Table

Table will read/write, select, modify, or show the information of the rows and columns in recognized Table formats (including FITS binary, FITS ASCII, and plain text table files, see Section 4.7 [Tables], page 183). Output columns can also be determined by number or regular expression matching of column names, units, or comments. The executable name is asttable with the following general template

```
$ asttable [OPTION...] InputFile
```

One line examples:

```
## Get the table column information (name, data type, or units):
$ asttable bintab.fits --information

## Print columns named RA and DEC, followed by all the columns where ## the name starts with "MAG_":
$ asttable bintab.fits --column=RA --column=DEC --column=/^MAG_/`
## Similar to the above, but with one call to `--column` (or `--c`),
## also sort the rows by the input's photometric redshift (`Z_PHOT`) column. To confirm the sort, you can add `Z_PHOT` to the columns to print.
$ asttable bintab.fits -cRA,DEC,/^MAG_/ --sort=Z_PHOT

## Similar to the above, but only print rows that have a photometric redshift between 2 and 3.
$ asttable bintab.fits -cRA,DEC,/^MAG_/ --range=Z_PHOT,2:3

## Only print rows with a value in the 10th column above 100000:
$ asttable bintab.fits --range=10,10e5,inf

## Only print the 2nd column, and the third column multiplied by 5, save the resulting two columns in `table.txt`
$ asttable bintab.fits -c2,'arith $2 5 x' -o table.txt

## Sort the output columns by the third column, save output:
$ asttable bintab.fits --sort=3 -o output.txt

## Subtract the first column from the second in `cat.fits` (can also be a text table) and keep the third and fourth columns.
$ asttable cat.txt -c'arith $2 $1 -',3,4 -o cat.fits

## Convert sexagesimal coordinates to degrees (same can be done in a large table given as argument).
$ echo "7h34m35.5498 31d53m14.352s" | asttable

## Convert RA and Dec in degrees to sexagesimal (same can be done in a large table given as argument).
echo "113.64812416667 31.88732" \ 
| asttable -c'arith $1 degree-to-ra $2 degree-to-dec'

Table’s input dataset can be given either as a file or from Standard input (piped from another program, see Section 4.1.4 [Standard input], page 167). In the absence of selected columns, all the input’s columns and rows will be written to the output. The full set of operations Table can do are described in detail below, but for a more high-level introduction to the various operations, and their precedence, see Section 5.3.2 [Operation precedence in Table], page 235.

If any output file is explicitly requested (with `--output`) the output table will be written in it. When no output file is explicitly requested the output table will be written to the standard output. If the specified output is a FITS file, the type of FITS table (binary or ASCII) will be determined from the `--tabletype` option. If the output is not a FITS file, it will be printed as a plain text table (with space characters between the columns). When the columns are accompanied by meta-data (like column name, units, or comments), this information will also printed in the plain text file before the table, as described in Section 4.7.2 [Gnuastro text table format], page 185.
For the full list of options common to all Gnuastro programs please see Section 4.1.2 [Common options], page 156. Options can also be stored in directory, user or system-wide configuration files to avoid repeating on the command-line, see Section 4.2 [Configuration files], page 169. Table does not follow Automatic output that is common in most Gnuastro programs, see Section 4.9 [Automatic output], page 191. Thus, in the absence of an output file, the selected columns will be printed on the command-line with no column information, ready for redirecting to other tools like awk.

**Sexagesimal coordinates as floats in plain-text tables:** When a column is determined to be a floating point type (32-bit or 64-bit) in a plain-text table, it can contain sexagesimal values in the format of ‘_h_m_s’ (for RA) and ‘_d_m_s’ (for Dec), where the ‘_’s are place-holders for numbers. In this case, the string will be immediately converted to a single floating point number (in units of degrees) and stored in memory with the rest of the column or table. Besides being useful in large tables, with this feature, conversion to sexagesimal coordinates to degrees becomes very easy, for example:

```bash
echo "7h34m35.5498 31d53m14.352s" | asttable
```

The inverse can also be done with the more general column arithmetic operators:

```bash
echo "113.64812416667 31.88732" \
| asttable -c'arith $1 degree-to-ra $2 degree-to-dec'
```

If you want to preserve the sexagesimal contents of a column, you should store that column as a string, see Section 4.7.2 [Gnuastro text table format], page 185.

- **-i**
  --**information**

Only print the column information in the specified table on the command-line and exit. Each column’s information (number, name, units, data type, and comments) will be printed as a row on the command-line. Note that the FITS standard only requires the data type (see Section 4.5 [Numeric data types], page 178), and in plain text tables, no meta-data/information is mandatory. Gnuastro has its own convention in the comments of a plain text table to store and transfer this information as described in Section 4.7.2 [Gnuastro text table format], page 185.

This option will take precedence over all other operations in Table, so when it is called along with other operations, they will be ignored, see Section 5.3.2 [Operation precedence in Table], page 235. This can be useful if you forget the identifier of a column after you have already typed some on the command-line. You can simply add a `-i` to your already-written command (without changing anything) and run Table, to see the whole list of column names and information. Then you can use the shell history (with the up arrow key on the keyboard), and retrieve the last command with all the previously typed columns present, delete `-i` and add the identifier you had forgot.

- **-c STR/INT**
  --**column=STR/INT**

Set the output columns either by specifying the column number, or name. For more on selecting columns, see Section 4.7.3 [Selecting table columns], page 188.
If a value of this option starts with `arith`, column arithmetic will be activated, allowing you to edit/manipulate column contents. For more on column arithmetic see Section 5.3.1 [Column arithmetic], page 230.

To ask for multiple columns this option can be used in two ways: 1) multiple calls to this option, 2) using a comma between each column specifier in one call to this option. These different solutions may be mixed in one call to Table: for example, `"-cRA,DEC,MAG"` or `"-cRA,DEC -cMAG"` are both equivalent to `"-cRA -cDEC -cMAG"`. The order of the output columns will be the same order given to the option or in the configuration files (see Section 4.2.2 [Configuration file precedence], page 170).

This option is not mandatory, if no specific columns are requested, all the input table columns are output. When this option is called multiple times, it is possible to output one column more than once.

```
-w FITS
--wcsfile=FITS
```

FITS file that contains the WCS to be used in the `wcs-to-img` and `img-to-wcs` operators of Section 5.3.1 [Column arithmetic], page 230. The extension name/number within the FITS file can be specified with `--wcshdu`.

If the value to this option is `none`, no WCS will be written in the output.

```
-W STR
--wcshdu=STR
```

FITS extension/HDU in the FITS file given to `--wcsfile` (see the description of `--wcsfile` for more).

```
-L FITS/TXT
--catcolumnfile=FITS/TXT
```

Concatenate (or add, or append) the columns of this option’s value (a filename) to the output columns. This option may be called multiple times (to add columns from more than one file into the final output), the columns from each file will be added in the same order that this option is called. The number of rows in the file(s) given to this option has to be the same as the input table (before any type of row-selection), see Section 5.3.2 [Operation precedence in Table], page 235.

By default all the columns of the given file will be appended, if you only want certain columns to be appended, use the `--catcolumns` option to specify their name or number (see Section 4.7.3 [Selecting table columns], page 188). Note that the columns given to `--catcolumns` must be present in all the given files (if this option is called more than once with more than one file).

If the file given to this option is a FITS file, its necessary to also define the corresponding HDU/extension with `--catcolumnhdu`. Also note that no operation (for example row selection, arithmetic or etc) is applied to the table given to this option.

If the appended columns have a name, the column names of each file will be appended with a `-N`, where `N` is a counter starting from 1 for each appended file.

This is done because when concatenating columns from multiple tables (more
than two) into one, they may have the same name, and its not good practice to have multiple columns with the same name. You can disable this feature with --catcolumnrawname. Generally, you can use the --colmetadata option to update column metadata.

For example, let’s assume you have two catalogs of the same objects (same number of rows) in different filters. Such that f160w-cat.fits has a MAGNITUDE column that has the magnitude of each object in the F160W filter and similarly f105w-cat.fits, also has a MAGNITUDE column, but for the F105W filter. You can use column concatenation like below to import the MAGNITUDE column from the F105W catalog into the F160W catalog, while giving each magnitude column a different name:

```bash
asttable f160w-cat.fits --output=both.fits \
--output=both.fits --catcolumnfile=f105w-cat.fits --catcolumns=MAGNITUDE \
--colmetadata=MAGNITUDE,MAG-F160W,log,"Magnitude in F160W" \
--colmetadata=MAGNITUDE-1,MAG-F105W,log,"Magnitude in F105W"
```

For a more complete example, see Section 2.2.15 [Working with catalogs (estimating colors)], page 58.

-u STR/INT
--catcolumnhdul=STR/INT
The HDU/extension of the FITS file(s) that should be concatenated, or appended, by column with --catcolumnfile. If --catcolumn is called more than once with more than one FITS file, its necessary to call this option more than once. The HDUs will be loaded in the same order as the FITS files given to --catcolumnfile.

-C STR/INT
--catcolumns=STR/INT
The column(s) in the file(s) given to --catcolumnfile to append. When this option is not given, all the columns will be concatenated. See --catcolumnfile for more.

--catcolumnrawname
Don’t modify the names of the concatenated (appended) columns, see description in --catcolumnfile.

-R FITS/TXT
--catrowfile=FITS/TXT
Add the rows of the given file to the output table. The selected columns in the tables given to this option should have the same number and datatype and the rows before control reaches this phase (after column selection and column concatenation), for more see Section 5.3.2 [Operation precedence in Table], page 235.

For example, if a.fits, b.fits and c.fits have the columns RA, DEC and MAGNITUDE (possibly in different column-numbers in their respective table, along with many more columns), the command below will add their rows into the final output that will only have these three columns:

```bash
$ asttable a.fits --catrowfile=b.fits --catrowhdu=1 \
```
Chapter 5: Data containers

```
--catrowfile=c.fits --catrowhdu=1 \
-cRA,DEC,MAGNITUDE --output=allrows.fits
```

**How to avoid repetition when adding rows:** this option will simply add the rows of multiple tables into one, it doesn’t check their contents! Therefore if you use this option on multiple catalogs that may have some shared physical objects in some of their rows, those rows/objects will be repeated in the final table. In such scenarios, to avoid potential repetition, it is better to use Section 7.5 [Match], page 429, (with `--notmatched` and `--outcols=AAA,BBB`) instead of Table. For more on using Match for this scenario, see the description of `--outcols` in Section 7.5.2 [Invoking Match], page 432.

**-X STR**

**--catrowhdu=STR**

The HDU/extension of the FITS file(s) that should be concatenated, or appended, by rows with `--catrowfile`. If `--catrowfile` is called more than once with more than one FITS file, its necessary to call this option more than once also (once for every FITS table given to `--catrowfile`). The HDUs will be loaded in the same order as the FITS files given to `--catrowfile`.

**-O --colinfoinstdout**

Add column metadata when the output is printed in the standard output. Usually the standard output is used for a fast visual check, or to pipe into other metadata-agnostic programs (like AWK) for further processing. So by default meta-data aren’t included. But when piping to other Gnuastro programs (where metadata can be interpreted and used) it is recommended to use this option and use column names in the next program.

**-r STR,FLT:FLT**

**--range=STR,FLT:FLT**

Only output rows that have a value within the given range in the STR column (can be a name or counter). Note that the range is only inclusive in the lower-limit. For example with `--range=sn,5:20` the output’s columns will only contain rows that have a value in the sn column (not case-sensitive) that is greater or equal to 5, and less than 20. Also you can use the comma for separating the values such as this `--range=sn,5,20`. For the precedence of this operation in relation to others, see Section 5.3.2 [Operation precedence in Table], page 235.

This option can be called multiple times (different ranges for different columns) in one run of the Table program. This is very useful for selecting the final rows from multiple criteria/columns.

The chosen column doesn’t have to be in the output columns. This is good when you just want to select using one column’s values, but don’t need that column anymore afterwards.

For one example of using this option, see the example under `--sigclip-median` in Section 7.1.5 [Invoking Statistics], page 350.
--inpolygon=STR1,STR2
Only return rows where the given coordinates are inside the polygon specified by the --polygon option. The coordinate columns are the given STR1 and STR2 columns, they can be a column name or counter (see Section 4.7.3 [Selecting table columns], page 188). For the precedence of this operation in relation to others, see Section 5.3.2 [Operation precedence in Table], page 235.

Note that the chosen columns doesn’t have to be in the output columns (which are specified by the --column option). For example if we want to select rows in the polygon specified in Section 2.2.4 [Dataset inspection and cropping], page 32, this option can be used like this (you can remove the double quotations and write them all in one line if you remove the white-spaces around the colon separating the column vertices):

```
asttable table.fits --inpolygon=RA,DEC
  --polygon="53.187414,-27.779152
  : 53.159507,-27.759633
  : 53.134517,-27.787144
  : 53.161906,-27.807208"
```

Flat/Euclidean space: The --inpolygon option assumes a flat/Euclidean space so it is only correct for RA and Dec when the polygon size is very small like the example above. If your polygon is a degree or larger, it may not return correct results. Please get in touch if you need such a feature (see Section 1.9 [Suggest new feature], page 13).

--outpolygon=STR1,STR2
Only return rows where the given coordinates are outside the polygon specified by the --polygon option. This option is very similar to the --inpolygon option, so see the description there for more.

--polygon=STR
--polygon=FLT,FLT:FLT,FLT:...
The polygon to use for the --inpolygon and --outpolygon options. This option is parsed in an identical way to the same option in the Crop program, so for more information on how to use it, see Section 6.1.4.1 [Crop options], page 264.

-e STR,INT/FLT,...
--equal=STR,INT/FLT,...
Only output rows that are equal to the given number(s) in the given column. The first argument is the column identifier (name or number, see Section 4.7.3 [Selecting table columns], page 188), after that you can specify any number of values. For the precedence of this operation in relation to others, see Section 5.3.2 [Operation precedence in Table], page 235.

For example --equal=ID,5,6,8 will only print the rows that have a value of 5, 6, or 8 in the ID column. This option can also be called multiple times, so --equal=ID,4,5 --equal=ID,6,7 has the same effect as --equal=4,5,6,7.
Equality and floating point numbers: Floating point numbers are only approximate values (see Section 4.5 [Numeric data types], page 178). In this context, their equality depends on how the input table was originally stored (as a plain text table or as an ASCII/binary FITS table). If you want to select floating point numbers, it is strongly recommended to use the --range option and set a very small interval around your desired number, don't use --equal or --notequal.

The --equal and --notequal options also work when the given column has a string type. In this case the given value to the option will also be parsed as a string, not as a number. When dealing with string columns, be careful with trailing white space characters (the actual value maybe adjusted to the right, left, or center of the column's width). If you need to account for such white spaces, you can use shell quoting. For example --equal=NAME," myname ".

Strings with a comma (,): When your desired column values contain a comma, you need to put a '\' before the internal comma (within the value). Otherwise, the comma will be interpreted as a delimiter between multiple values, and anything after it will be interpreted as a separate string. For example, assume column AB of your table.fits contains this value: 'cd,ef' in your desired rows. To extract those rows, you should use the command below:

```
$ asttable table.fits --equal=AB,cd\,ef
```

-n STR,INT/FLT,...
--notequal=STR,INT/FLT,...

Only output rows that are not equal to the given number(s) in the given column. The first argument is the column identifier (name or number, see Section 4.7.3 [Selecting table columns], page 188), after that you can specify any number of values. For example --notequal=ID,5,6,8 will only print the rows where the ID column doesn’t have value of 5, 6, or 8. This option can also be called multiple times, so --notequal=ID,4,5 --notequal=ID,6,7 has the same effect as --notequal=4,5,6,7.

Be very careful if you want to use the non-equality with floating point numbers, see the special note under --equal for more. This option also works when the given column has a string type, see the description under --equal (above) for more.

-b STR[,STR[,STR]]
--noblank=STR[,STR[,STR]]

Only output rows that are not blank in the given column of the input table. Like above, the columns can be specified by their name or number (counting from 1). This option can be called multiple times, so --noblank=MAG --noblank=PHOTOZ is equivalent to --noblank=MAG,PHOTOZ. For the precedence of this operation in relation to others, see Section 5.3.2 [Operation precedence in Table], page 235.

For example if table.fits has blank values (NaN in floating point types) in the magnitude and sn columns, with --noblank=magnitude,sn, the output will not contain any rows with blank values in these two columns.
If you want all columns to be checked, simply set the value to \_all (in other words: \_noblank=\_all). This mode is useful when there are many columns in the table and you want a “clean” output table (with no blank values in any column): entering their name or number one-by-one can be buggy and frustrating. In this mode, no other column name should be given. For example if you give \_noblank=\_all,magnitude, then Table will assume that your table actually has a column named \_all and magnitude, and if it doesn’t, it will abort with an error.

If you want to change column values using Section 5.3.1 [Column arithmetic], page 230, (and set some to blank, to later remove), or you want to select rows based on columns that you have imported from other tables, you should use the \_noblankend option described below. Also, see Section 5.3.2 [Operation precedence in Table], page 235.

\-s STR
\--sort=STR
Sort the output rows based on the values in the STR column (can be a column name or number). By default the sort is done in ascending/increasing order, to sort in a descending order, use \--descending. For the precedence of this operation in relation to others, see Section 5.3.2 [Operation precedence in Table], page 235.

The chosen column doesn’t have to be in the output columns. This is good when you just want to sort using one column’s values, but don’t need that column anymore afterwards.

\-d
\--descending
When called with \--sort, rows will be sorted in descending order.

\-H INT
\--head=INT
Only print the given number of rows from the top of the final table. Note that this option only affects the output table. For example if you use \--sort, or \--range, the printed rows are the first after applying the sort sorting, or selecting a range of the full input. This option cannot be called with \--tail, \--rowlimit or \--rowrandom. For the precedence of this operation in relation to others, see Section 5.3.2 [Operation precedence in Table], page 235.

If the given value to \--head is 0, the output columns won’t have any rows and if its larger than the number of rows in the input table, all the rows are printed (this option is effectively ignored). This behavior is taken from the head program in GNU Coreutils.

\-t INT
\--tail=INT
Only print the given number of rows from the bottom of the final table. See \--head for more. This option cannot be called with \--head, \--rowlimit or \--rowrandom.
--rowlimit=INT,INT
Only return the rows within the requested positional range (inclusive on both
sides). Therefore, `--rowlimit=5,7` will return 3 of the input rows, row 5, 6
and 7. This option will abort if any of the given values is larger than the total
number of rows in the table. For the precedence of this operation in relation to
others, see Section 5.3.2 [Operation precedence in Table], page 235.
With the `--head` or `--tail` options you can only see the top or bottom few rows.
However, with this option, you can limit the returned rows to a contiguous set
of rows in the middle of the table. Therefore this option cannot be called with
`--head`, `--tail`, or `--rowlimit`.

--rowrandom=INT
Select INT rows from the input table by random (assuming a uniform distri-
bution). This option is applied after the value-based selection options (like
`--sort`, `--range`, `--polygon` and etc). On the other hand, only the row coun-
ters are randomly selected, this option doesn’t change the order. Therefore, if
`--rowrandom` is called together with `--sort`, the returned rows are still sorted.
This option cannot be called with `--head`, `--tail`, or `--rowlimit`. For the
precedence of this operation in relation to others, see Section 5.3.2 [Operation
precedence in Table], page 235.
This option will only have an effect if INT is larger than the number of rows
when it is activated (after the value-based selection options have been applied).
When there are fewer rows, a warning is printed, saying that this option has no
effect. The warning can be disabled with the `--quiet` option.
Due to its nature (to be random), the output of this option differs in each
run. Therefore 5 calls to Table with `--rowrandom` on the same input table will
generate 5 different outputs. If you want a reproducible random selection, set
the GSL_RNG_SEED environment variable and also use the `--envseed` option, for
more see Section 8.2.1.4 [Generating random numbers], page 464.

--envseed
Read the random number generator seed from the GSL_RNG_SEED environment
variable for `--rowrandom` (instead of generating a different seed internally on ev-
every run). This is useful if you want a reproducible random selection of the input
rows. For more, see Section 8.2.1.4 [Generating random numbers], page 464.

-E STR[,STR[,STR]]
--noblankend=STR[,STR[,STR]]
Remove all rows in the requested output columns that have a blank value. Like
above, the columns can be specified by their name or number (counting from 1).
This option can be called multiple times, so `--noblank=MAG --noblank=PHOTOZ`
is equivalent to `--noblank=MAG,PHOTOZ`. For the precedence of this operation in
relation to others, see Section 5.3.2 [Operation precedence in Table], page 235.
For example if your final output table (possibly after column arithmetic, or
adding new columns) has blank values (NaN in floating point types) in the
magnitude and sn columns, with `--noblankend=magnitude,sn`, the output
will not contain any rows with blank values in these two columns.
If you want blank values to be removed from the main input table _before_ any further processing (like adding columns, sorting or column arithmetic), you should use the **--noblank** option. With the **--noblank** option, the column(s) that is(are) given doesn’t necessarily have to be in the output (it is just temporarily used for reading the inputs and selecting rows, but doesn’t necessarily need to be present in the output). However, the column(s) given to this option should exist in the output.

If you want _all_ columns to be checked, simply set the value to **_all**** (in other words: **--noblankend=_all**). This mode is useful when there are many columns in the table and you want a “clean” output table (with no blank values in any column): entering their name or number one-by-one can be buggy and frustrating. In this mode, no other column name should be given. For example if you give **--noblankend=_all,magnitude**, then Table will assume that your table actually has a column named _all and magnitude, and if it doesn’t, it will abort with an error.

This option is applied just before writing the final table (after **--colmetadata** has finished). So in case you changed the column metadata, or added new columns, you can use the new names, or the newly defined column numbers. For the precedence of this operation in relation to others, see Section 5.3.2 [Operation precedence in Table], page 235.

**-m STR/INT,STR[,STR[,STR]]**

**--colmetadata=STR/INT,STR[,STR[,STR]]**

Update the specified column metadata in the output table. This option is applied after all other column-related operations are complete, for example column arithmetic, or column concatenation. For the precedence of this operation in relation to others, see Section 5.3.2 [Operation precedence in Table], page 235.

The first value (before the first comma) given to this option is the column’s identifier. It can either be a counter (positive integer, counting from 1), or a name (the column’s name in the output if this option wasn’t called).

After the to-be-updated column is identified, at least one other string should be given, with a maximum of three strings. The first string after the original name will the selected column’s new name. The next (optional) string will be the selected column’s unit and the third (optional) will be its comments. If the two optional strings aren’t given, the original column’s units or comments will remain unchanged.

If any of the values contains a comma, you should place a `\` before the comma to avoid it getting confused with a delimiter. For example see the command below for a column description that contains a comma:

```
$ asttable table.fits \
    --colmetadata=NAME,UNIT,"Comments\\, with a comma"
```

Generally, since the comma is commonly used as a delimiter in many scenarios, to avoid complicating your future analysis with the table, it is best to avoid using a comma in the column name and units.
Some examples of this option are available in the tutorials, in particular Section 2.2.15 [Working with catalogs (estimating colors)], page 58. Here are some more specific examples:

```bash
--colmetadata=MAGNITUDE,MAG_F160W
```
This will convert name of the original MAGNITUDE column to MAG_F160W, leaving the unit and comments unchanged.

```bash
--colmetadata=3,MAG_F160W,mag
```
This will convert name of the third column of the final output to MAG_F160W and the units to mag, while leaving the comments untouched.

```bash
--colmetadata=MAGNITUDE,MAG_F160W,mag,"Magnitude in F160W filter"
```
This will convert name of the original MAGNITUDE column to MAG_F160W, and the units to mag and the comments to Magnitude in F160W filter. Note the double quotations around the comment string, they are necessary to preserve the white-space characters within the column comment from the command-line, into the program (otherwise, upon reaching a white-space character, the shell will consider this option to be finished and cause unexpected behavior).

If your table is large and generated by a script, you can first do all your operations on your table’s data and write it into a temporary file (maybe called temp.fits). Then, look into that file’s metadata (with asttable temp.fits -i) to see the exact column positions and possible names, then add the necessary calls to this option to your previous call to asttable, so it writes proper metadata in the same run (for example in a script or Makefile). Recall that when a name is given, this option will update the metadata of the first column that matches, so if you have multiple columns with the same name, you can call this options multiple times with the same first argument to change them all to different names.

Finally, if you already have a FITS table by other means (for example by downloading) and you merely want to update the column metadata and leave the data intact, it is much more efficient to directly modify the respective FITS header keywords with astfits, using the keyword manipulation features described in Section 5.1.1.2 [Keyword inspection and manipulation], page 201. --colmetadata is mainly intended for scenarios where you want to edit the data so it will always load the full/partial dataset into memory, then write out the resulting datasets with updated/corrected metadata.

### 5.4 Query

There are many astronomical databases available for downloading astronomical data. Most follow the International Virtual Observatory Alliance (IVOA, [https://ivoa.net](https://ivoa.net)) standards (and in particular the Table Access Protocol, or TAP\(^\text{16}\)). With TAP, it is possible to

\(^{16}\text{https://ivoa.net/documents/TAP}\)
submit your queries via a command-line downloader (for example `curl`) to only get specific tables, targets (rows in a table) or measurements (columns in a table): you don’t have to download the full table (which can be very large in some cases)! These customizations are done through the Astronomical Data Query Language (ADQL\(^\text{17}\)).

Therefore, if you are sufficiently familiar with TAP and ADQL, you can easily custom-download any part of an online dataset. However, you also need to keep a record of the URLs of each database and in many cases, the commands will become long and hard/buggy to type on the command-line. On the other hand, most astronomers don’t know TAP or ADQL at all, and are forced to go to the database’s web page which is slow (it needs to download so many images, and has too much annoying information), requires manual interaction (further making it slow and buggy), and can’t be automated.

Gnuastro’s Query program is designed to be the middle-man in this process: it provides a simple high-level interface to let you specify your constraints on what you want to download. It then internally constructs the command to download the data based on your inputs and runs it to download your desired data. Query also prints the full command before it executes it (if not called with `--quiet`). Also, if you ask for a FITS output table, the full command is written into its 0-th extension along with other input parameters to query (all Gnuastro programs generally keep their input configuration parameters as FITS keywords in the zero-th output). You can see it with Gnuastro’s Fits program, like below:

```
$ astfits query-output.fits -h0
```

With the full command used to download the dataset, you only need a minimal knowledge of ADQL to do lower-level customizations on your downloaded dataset. You can simply copy that command and change the parts of the query string you want: ADQL is very powerful! For example you can ask the server to do mathematical operations on the columns and apply selections after those operations, or combine/match multiple datasets and etc. We will try to add high-level interfaces for such capabilities, but generally, don’t limit yourself to the high-level operations (that can’t cover everything!).

### 5.4.1 Available databases

The current list of databases supported by Query are listed at the end of this section. To get the list of available datasets within each database, you can use the `--information` option. For example with the command below you can get a list of the roughly 100 datasets that are available within the ESA Gaia server with their description:

```
$ astquery gaia --information
```

However, other databases like VizieR host many more datasets (tens of thousands!). Therefore it is very inconvenient to get the full information every time you want to find your dataset of interest (the full metadata file VizieR is more than 20Mb). In such cases, you can limit the downloaded and displayed information with the `--limitinfo` option. For example with the first command below, you can get all datasets relating to the MUSE (an instrument on the Very Large Telescope), and those that include Roland Bacon (Principal Investigator of MUSE) as an author (Bacon, R.). Recall that `-i` is the short format of `--information`.

```
$ astquery vizier -i --limitinfo=MUSE
```

\(^17\) [https://ivoa.net/documents/ADQL](https://ivoa.net/documents/ADQL)
$ astquery vizier -i --limitinfo="Bacon R."

Once you find the recognized name of your desired dataset, you can see the column information of that dataset with adding the dataset name. For example, with the command below you can see the column metadata in the J/A+A/608/A2/udf10 dataset (one of the datasets in the search above) using this command:

$ astquery vizier --dataset=J/A+A/608/A2/udf10 -i

For very popular datasets of a database, Query provides an easier-to-remember short name that you can feed to --dataset. This short name will map to the officially recognized name of the dataset on the server. In this mode, Query will also set positional columns accordingly. For example most VizieR datasets have an RAJ2000 column (the RA and the epoch of 2000) so it is the default RA column name for coordinate search (using --center or --overlapwith). However, some datasets don’t have this column (for example SDSS DR12). So when you use the short name and Query knows about this dataset, it will internally set the coordinate columns that SDSS DR12 has: RA_ICRS and DEC_ICRS. Recall that you can always change the coordinate columns with --ccol.

For example in the VizieR and Gaia databases, the recognized name for the early data release 3 data is respectively I/350/gaiaedr3 and gaiaedr3.gaia_source. These technical names can be hard to remember. Therefore Query provides gaiaedr3 (for VizieR) and edr3 (for ESA’s Gaia) shortcuts which you can give to --dataset instead. They will be directly mapped to the fully recognized name by Query. In the list below that describes the available databases, the available short names are also listed.

---

**Not all datasets support TAP:** Large databases like VizieR have TAP access for all their datasets. However, smaller databases haven’t implemented TAP for all their tables. Therefore some datasets that are searchable in their web interface may not be available for a TAP search. To see the full list of TAP-ed datasets in a database, use the --information (or -i) option with the dataset name like the command below.

$ astquery astron -i

If your desired dataset isn’t in this list, but has web-access, contact the database maintainers and ask them to add TAP access for it. After they do it, you should see the name added to the output list of the command above.

---

The list of databases recognized by Query (and their names in Query) is described below. Since Query is a new member of the Gnuastro family (first available in Gnuastro 0.14), this list will hopefully grow significantly in the next releases. If you have any particular datasets in mind, please let us know by sending an email to bug-gnuastro@gnu.org. If the dataset supports IVOA’s TAP (Table Access Protocol), it should be very easy to add.

**astron**

The ASTRON Virtual Observatory service ([https://vo.astron.nl](https://vo.astron.nl)) is a database focused on radio astronomy data and images, primarily those collected by ASTRON itself. A query to astron is submitted to [https://vo.astron.nl/_system_/tap/run/tap-sync](https://vo.astron.nl/_system_/tap/run/tap-sync).

Here is the list of short names for dataset(s) in ASTRON’s VO service:

- tgssadr --> tgssadr.main
gaia

The Gaia project (https://www.cosmos.esa.int/web/gaia) database which is a large collection of star positions on the celestial sphere, as well as peculiar velocities, parallaxes and magnitudes in some bands among many others. Besides scientific studies (like studying resolved stellar populations in the Galaxy and its halo), Gaia is also invaluable for raw data calibrations, like astrometry. A query to gaia is submitted to https://gea.esac.esa.int/tap-server/tap/sync.

Here is the list of short names for popular datasets within Gaia:

- edr3 --> gaiaedr3.gaia_source
- dr2 --> gaiadr2.gaia_source
- dr1 --> gaiadr1.gaia_source
- tycho2 --> public.tycho2
- hipparcos --> public.hipparcos

ned

The NASA/IPAC Extragalactic Database (NED, http://ned.ipac.caltech.edu) is a fusion database, integrating the information about extra-galactic sources from many large sky surveys into a single catalog. It covers the full spectrum, from Gamma rays to radio frequencies and is updated when new data arrives. A TAP query to ned is submitted to https://ned.ipac.caltech.edu/tap/sync.

- objdir --> NEDTAP.objdir: default TAP-based dataset in NED.
- extinction: A command-line interface to the NED Extinction Calculator (https://ned.ipac.caltech.edu/extinction_calculator). It only takes a central coordinate and returns a VOTable of the calculated extinction in many commonly used filters at that point. As a result, options like --width or --radius are not supported. However, Gnuastro doesn’t yet support the VOTable format. Therefore, if you specify an --output file, it should have an .xml suffix and the downloaded file will not be checked.

Until VOTable support is added to Gnuastro, you can use GREP, AWK and SED to convert the VOTable data into a FITS table with a command like below (assuming the queried VOTable is called ned-extinction.xml):

```
grep '^-<TR><TD>' ned-extinction.xml \
| sed -e's|<TR><TD>||' \
  -e's</TD>|/\' \
  -e's</TD><TD>|@|g' \
| awk 'BEGIN{FS="@"};' \
  print "# Column 1: FILTER [name,str15] Filter name"; \
  print "# Column 2: CENTRAL [um,f32] Central Wavelength"; \
  print "# Column 3: EXTINCTION [mag,f32] Galactic Ext."; \
  print "# Column 4: ADS_REF [ref,str50] ADS reference"} \
  {printf "%-15s %g %g %s\n", $1, $2, $3, $4}' \
| asttable -oned-extinction.fits
```

Once the table is in FITS, you can easily get the extinction for a certain filter (for example the SDSS r filter) like the command below:

```
asttable ned-extinction.fits --equal=FILTER,"SDSS r" \
-cEXTINCTION
```
Vizier (https://vizier.u-strasbg.fr) is arguably the largest catalog database in astronomy, containing more than 20500 catalogs as of mid January 2021. Almost all published catalogs in major projects, and even the tables in many papers are archived and accessible here. For example, VizieR also has a full copy of the Gaia database mentioned below, with some additional standardized columns (like RA and Dec in J2000).

The current implementation of --limitinfo only looks into the description of the datasets, but since VizieR is so large, there is still a lot of room for improvement. Until then, if --limitinfo isn’t sufficient, you can use VizieR’s own web-based search for your desired dataset: http://cdsarc.u-strasbg.fr/viz-bin/cat

Because VizieR curates such a diverse set of data from tens of thousands of projects and aims for interoperability between them, the column names in VizieR may not be identical to the column names in the surveys’ own databases (Gaia in the example above). A query to vizier is submitted to http://tapvizier.u-strasbg.fr/TAPVizieR/tap/sync.

Here is the list of short names for popular datasets within VizieR (sorted alphabetically by their short name). Please feel free to suggest other major catalogs (covering a wide area or commonly used in your field). For details on each dataset with necessary citations, and links to web pages, look into their details with their ViziR names in https://vizier.u-strasbg.fr/viz-bin/VizieR.

- 2mass --> II/246/out (2MASS All-Sky Catalog)
- akarifis --> II/298/fis (AKARI/FIS All-Sky Survey)
- allwise --> II/328/allwise (AllWISE Data Release)
- apass9 --> II/336/apass9 (AAVSO Photometric All Sky Survey, DR9)
- catwise --> II/365/catwise (CatWISE 2020 catalog)
- des1 --> II/357/des_dr1 (Dark Energy Survey data release 1)
- gaiadr2 --> I/345/gaia2 (GAIA Data Release 2)
- gaiadrd3 --> I/350/gaiadrd3 (GAIA early Data Release 3)
- galex5 --> II/312/ais (All-sky Survey of GALEX DR5)
- nomad --> I/297/out (Naval Observatory Merged Astrometric Dataset)
- panstarrs1 --> II/349/ps1 (Pan-STARRS Data Release 1)
- ppmx1 --> I/317/sample (Positions and proper motions on the ICRS)
- sdss12 --> V/147/sdss12 (SDSS Photometric Catalogue, Release 12)
- usnob1 --> I/284/out (Whole-Sky USNO-B1.0 Catalog)
- ucac5 --> I/340/ucac5 (5th U.S. Naval Obs. CCD Astrograph Catalog)
- unwise --> II/363/unwise (Band-merged unWISE Catalog)
- wise --> II/311/wise (WISE All-Sky data Release)

5.4.2 Invoking Query

Query provides a high-level interface to downloading subsets of data from databases. The executable name is astquery with the following general template

$ astquery DATABASE-NAME [OPTION...] ...
One line examples:

```bash
## Information about all datasets in ESA's GAIA database:
$ astquery gaia --information

## Only show catalogs in VizieR that have 'MUSE' in their 
## description. The '-i' is short for '--information'.
$ astquery vizier -i --limitinfo=MUSE

## List of columns in 'J/A+A/608/A2/udf10' (one of the above).
$ astquery vizier --dataset=J/A+A/608/A2/udf10 -i

## ID, RA and Dec of all Gaia sources within an image.
$ astquery gaia --dataset=edr3 --overlapwith=image.fits \
   -csource_id,ra,dec

## RA, Dec and Spectroscopic redshifts of objects in SDSS DR12 
## spectroscopic redshift that overlap with 'image.fits'.
$ astquery vizier --dataset=sdss12 --overlapwith=image.fits \
   -cRA_ICRS,DE_ICRS,zsp --range=zsp,1e-10,inf

## All columns of all entries in the Gaia eDR3 catalog (hosted at 
## VizieR) within 1 arc-minute of the given coordinate.
$ astquery vizier --dataset=I/350/gaiaedr3 --output=my-gaia.fits \
   --center=113.8729761,31.9027152 --radius=1/60 \

## Similar to above, but only ID, RA and Dec columns for objects with 
## magnitude range 10 to 15. In VizieR, this column is called 'Gmag'.
## Also, using sexagesimal coordinates instead of degrees for center.
$ astquery vizier --dataset=I/350/gaiaedr3 --output=my-gaia.fits \
   --center=07h35m29.51,31d54m9.77 --radius=1/60 \
   --range=Gmag,10:15 -cEDR3Name,RAJ2000,DEJ2000
```

Query takes a single argument which is the name of the database. For the full list of available databases and accessing them, see Section 5.4.1 [Available databases], page 250. There are two methods to query the databases, each is more fully discussed in its option’s description below.

- **Low-level:** With `--query` you can directly give a raw query statement that is recognized by the database. This is very low level and will require a good knowledge of the database’s query language, but of course, it is much more powerful. If this option is given, the raw string is directly passed to the server and all other constraints/options (for Query’s high-level interface) are ignored.

- **High-level:** With the high-level options (like `--column`, `--center`, `--radius`, `--range` and other constraining options below), the low-level query will be constructed automatically for the particular database. This method is only limited to the generic capabilities that Query provides for all servers. So `--query` is more powerful, however, in this mode, you don’t need any knowledge of the database’s query language. You
can see the internally generated query on the terminal (if --quiet is not used) or in the 0-th extension of the output (if it is a FITS file). This full command contains the internally generated query.

The name of the downloaded output file can be set with --output. The requested output format can have any of the Section 4.7.1 [Recognized table formats], page 184, (currently .txt or .fits). Like all Gnuastro programs, if the output is a FITS file, the zero-th/first HDU of the output will contain all the command-line options given to Query as well as the full command used to access the server. When --output is not set, the output name will be in the format of NAME-STRING.fits, where NAME is the name of the database and STRING is a randomly selected 6-character set of numbers and alphabetic characters. With this feature, a second run of astquery that isn’t called with --output will not over-write an already downloaded one. Generally, when calling Query more than once, it is recommended to set an output name for each call based on your project’s context.

The outputs of Query will have a common output format, irrespective of the used database. To achieve this, Query will ask the databases to provide a FITS table output (for larger tables, FITS can consume much less download volume). After downloading is complete, the raw downloaded file will be read into memory once by Query, and written into the file given to --output. The raw downloaded file will be deleted by default, but can be preserved with the --keeprawdownload option. This strategy avoids unnecessary surprises depending on database. For example some databases can download a compressed FITS table, even though we ask for FITS. But with the strategy above, the final output will be an uncompressed FITS file. The metadata that is added by Query (including the full download command) is also very useful for future usage of the downloaded data. Unfortunately many databases don’t write the input queries into their generated tables.

--dry-run

Only print the final download command to contact the server, don’t actually run it. This option is good when you want to check the finally constructed query or download options given to the download program. You may also want to use the constructed command as a base to do furthercustomizations on it and run it yourself.

-k
--keeprawdownload

Don’t delete the raw downloaded file from the database. The name of the raw download will have a OUTPUT-raw-download.fits format. Where OUTPUT is either the base-name of the final output file (without a suffix).

-i
--information

Print the information of all datasets (tables) within a database or all columns within a database. When --dataset is specified, the latter mode (all column information) is downloaded and printed and when its not defined, all dataset information (within the database) is printed.

Some databases (like VizieR) contain tens of thousands of datasets, so you can limit the downloaded and printed information for available databases with the --limitinfo option (described below). Dataset descriptions are often large and contain a lot of text (unlike column descriptions). Therefore when printing the
information of all datasets within a database, the information (e.g., database name) will be printed on separate lines before the description. However, when printing column information, the output has the same format as a similar option in Table (see Section 5.3.3 [Invoking Table], page 238).

Important note to consider: the printed order of the datasets or columns is just for displaying in the printed output. You cannot ask for datasets or columns based on the printed order, you need to use dataset or column names.

-L STR  
--limitinfo=STR

Limit the information that is downloaded and displayed (with --information) to those that have the string given to this option in their description. Note that this is case-sensitive. This option is only relevant when --information is also called.

Databases may have thousands (or tens of thousands) of datasets. Therefore just the metadata (information) to show with --information can be tens of megabytes (for example the full VizieR metadata file is about 23Mb as of January 2021). Once downloaded, it can also be hard to parse manually. With --limitinfo, only the metadata of datasets that contain this string in their description will be downloaded and displayed, greatly improving the speed of finding your desired dataset.

-Q "STR"  
--query="STR"

Directly specify the query to be passed onto the database. The queries will generally contain space and other meta-characters, so we recommend placing the query within quotations.

-s STR  
--dataset=STR

The dataset to query within the database (not compatible with --query). This option is mandatory when --query or --information aren’t provided. You can see the list of available datasets within a database using --information (possibly supplemented by --limitinfo). The output of --information will contain the recognized name of the datasets within that database. You can pass the recognized name directly to this option. For more on finding and using your desired database, see Section 5.4.1 [Available databases], page 250.

-c STR  
--column=STR[,STR[,...]]

The column name(s) to retrieve from the dataset in the given order (not compatible with --query). If not given, all the dataset’s columns for the selected rows will be queried (which can be large!). This option can take multiple values in one instance (for example --column=ra,dec,mag), or in multiple instances (for example -cra -cdec -cmag), or mixed (for example -cra,dec -cmag).

In case, you don’t know the full list of the dataset’s column names a-priori, and you don’t want to download all the columns (which can greatly decrease your download speed), you can use the --information option combined with the --dataset option, see Section 5.4.1 [Available databases], page 250.
-H INT
--head=INT
Only ask for the first INT rows of the finally selected columns, not all the rows. This can be good when your search can result a large dataset, but before downloading the full volume, you want to see the top rows and get a feeling of what the whole dataset looks like.

-v FITS
--overlapwith=FITS
File name of FITS file containing an image (in the HDU given by --hdu) to use for identifying the region to query in the give database and dataset. Based on the image’s WCS and pixel size, the sky coverage of the image is estimated and values to the --center, --width will be calculated internally. Hence this option cannot be used with --center, --width or --radius. Also, since it internally generates the query, it can’t be used with --query.

Note that if the image has WCS distortions and the reference point for the WCS is not within the image, the WCS will not be well-defined. Therefore the resulting catalog may not overlap, or correspond to a larger/small area in the sky.

-C FLT,FLT
--center=FLT,FLT
The spatial center position (mostly RA and Dec) to use for the automatically generated query (not compatible with --query). The comma-separated values can either be in degrees (a single number), or sexagesimal (_h_m_ for RA, _d_m_ for Dec, or _:_:_ for both).

The given values will be compared to two columns in the database to find/return rows within a certain region around this center position will be requested and downloaded. Pre-defined RA and Dec column names are defined in Query for every database, however you can use --ccol to select other columns to use instead. The region can either be a circle and the point (configured with --radius) or a box/rectangle around the point (configured with --width).

--ccol=STR,STR
The name of the coordinate-columns in the dataset to compare with the values given to --center. Query will use its internal defaults for each dataset (for example RAJ2000 and DEJ2000 for VizieR data). But each dataset is treated separately and it isn’t guaranteed that these columns exist in all datasets. Also, more than one coordinate system/epoch may be present in a dataset and you can use this option to construct your spatial constraint based on the others coordinate systems/epochs.

-r FLT
--radius=FLT
The radius about the requested center to use for the automatically generated query (not compatible with --query). The radius is in units of degrees, but you can use simple division with this option directly on the command-line. For example if you want a radius of 20 arc-minutes or 20 arc-seconds, you can
use \texttt{--radius=20/60} or \texttt{--radius=20/3600} respectively (which is much more human-friendly than \(0.3333\) or \(0.005556\)).

\texttt{-w FLT[,FLT]}
\texttt{--width=FLT[,FLT]}

The square (or rectangle) side length (width) about the requested center to use for the automatically generated query (not compatible with \texttt{--query}). If only one value is given to \texttt{--width} the region will be a square, but if two values are given, the widths of the query box along each dimension will be different. The value(s) is (are) in the same units as the coordinate column (see \texttt{--ccol}, usually RA and Dec which are degrees). You can use simple division for each value directly on the command-line if you want relatively small (and more human-friendly) sizes. For example if you want your box to be 1 arc-minutes along the RA and 2 arc-minutes along Dec, you can use \texttt{--width=1/60,2/60}.

\texttt{-g STR,FLT,FLT}
\texttt{--range=STR,FLT,FLT}

The column name and numerical range (inclusive) of acceptable values in that column (not compatible with \texttt{--query}). This option can be called multiple times for applying range limits on many columns in one call (thus greatly reducing the download size). For example when used on the ESA gaia database, you can use \texttt{--range=phot\_g\_mean\_mag,10:15} to only get rows that have a value between 10 and 15 (inclusive on both sides) in the \texttt{phot\_g\_mean\_mag} column.

If you want all rows larger, or smaller, than a certain number, you can use \texttt{inf}, or \texttt{-inf} as the first or second values respectively. For example, if you want objects with SDSS spectroscopic redshifts larger than 2 (from the VizieR sdss12 database), you can use \texttt{--range=zsp,2,inf}

If you want the interval to not be inclusive on both sides, you can run \texttt{astquery} once and get the command that it executes. Then you can edit it to be non-inclusive on your desired side.

\texttt{-b STR[,STR]}
\texttt{--noblank=STR[,STR]}

Only ask for rows that don’t have a blank value in the \texttt{STR} column. This option can be called many times, and each call can have multiple column names (separated by a comma or ,). For example if you want the retrieved rows to not have a blank value in columns A, B, C and D, you can use \texttt{--noblank=A,-bB,C,D}.

\texttt{--sort=STR[,STR]}

Ask for the server to sort the downloaded data based on the given columns. For example let’s assume your desired catalog has column Z for redshift and column MAG\_R for magnitude in the R band. When you call \texttt{--sort=Z,MAG\_R}, it will primarily sort the columns based on the redshift, but if two objects have the same redshift, they will be sorted by magnitude. You can add as many columns as you like for higher-level sorting.
6 Data manipulation

Images are one of the major formats of data that is used in astronomy. The functions in this chapter explain the GNU Astronomy Utilities which are provided for their manipulation. For example cropping out a part of a larger image or convolving the image with a given kernel or applying a transformation to it.

6.1 Crop

Astronomical images are often very large, filled with thousands of galaxies. It often happens that you only want a section of the image, or you have a catalog of sources and you want to visually analyze them in small postage stamps. Crop is made to do all these things. When more than one crop is required, Crop will divide the crops between multiple threads to significantly reduce the run time.

Astronomical surveys are usually extremely large. So large in fact, that the whole survey will not fit into a reasonably sized file. Because of this, surveys usually cut the final image into separate tiles and store each tile in a file. For example the COSMOS survey’s Hubble space telescope, ACS F814W image consists of 81 separate FITS images, with each one having a volume of 1.7 Giga bytes.

Even though the tile sizes are chosen to be large enough that too many galaxies/targets don’t fall on the edges of the tiles, inevitably some do. So when you simply crop the image of such targets from one tile, you will miss a large area of the surrounding sky (which is essential in estimating the noise). Therefore in its WCS mode, Crop will stitch parts of the tiles that are relevant for a target (with the given width) from all the input images that cover that region into the output. Of course, the tiles have to be present in the list of input files.

Besides cropping postage stamps around certain coordinates, Crop can also crop arbitrary polygons from an image (or a set of tiles by stitching the relevant parts of different tiles within the polygon), see --polygon in Section 6.1.4 [Invoking Crop], page 263. Alternatively, it can crop out rectangular regions through the --section option from one image, see Section 6.1.2 [Crop section syntax], page 262.

6.1.1 Crop modes

In order to be comprehensive, intuitive, and easy to use, there are two ways to define the crop:

1. From its center and side length. For example if you already know the coordinates of an object and want to inspect it in an image or to generate postage stamps of a catalog containing many such coordinates.

2. The vertices of the crop region, this can be useful for larger crops over many targets, for example to crop out a uniformly deep, or contiguous, region of a large survey.

Irrespective of how the crop region is defined, the coordinates to define the crop can be in Image (pixel) or World Coordinate System (WCS) standards. All coordinates are read as floating point numbers (not integers, except for the --section option, see below). By setting the mode in Crop, you define the standard that the given coordinates must be interpreted. Here, the different ways to specify the crop region are discussed within each standard. For the full list options, please see Section 6.1.4 [Invoking Crop], page 263.
When the crop is defined by its center, the respective (integer) central pixel position will be found internally according to the FITS standard. To have this pixel positioned in the center of the cropped region, the final cropped region will have an add number of pixels (even if you give an even number to \texttt{--width} in image mode).

Furthermore, when the crop is defined as by its center, Crop allows you to only keep crops what don’t have any blank pixels in the vicinity of their center (your primary target). This can be very convenient when your input catalog/coordinates originated from another survey/filter which is not fully covered by your input image, to learn more about this feature, please see the description of the \texttt{--checkcenter} option in Section 6.1.4 [Invoking Crop], page 263.

Image coordinates

In image mode (\texttt{--mode=img}), Crop interprets the pixel coordinates and widths in units of the input data-elements (for example pixels in an image, not world coordinates). In image mode, only one image may be input. The output crop(s) can be defined in multiple ways as listed below.

Center of multiple crops (in a catalog)

The center of (possibly multiple) crops are read from a text file. In this mode, the columns identified with the \texttt{--coordcol} option are interpreted as the center of a crop with a width of \texttt{--width} pixels along each dimension. The columns can contain any floating point value. The value to \texttt{--output} option is seen as a directory which will host (the possibly multiple) separate crop files, see Section 6.1.4.2 [Crop output], page 270, for more. For a tutorial using this feature, please see Section 2.2.19 [Finding reddest clumps and visual inspection], page 68.

Center of a single crop (on the command-line)

The center of the crop is given on the command-line with the \texttt{--center} option. The crop width is specified by the \texttt{--width} option along each dimension. The given coordinates and width can be any floating point number.

Vertices of a single crop

In image mode there are two options to define the vertices of a region to crop: \texttt{--section} and \texttt{--polygon}. The former is lower-level (doesn’t accept floating point vertices, and only a rectangular region can be defined), it is also only available in Image mode. Please see Section 6.1.2 [Crop section syntax], page 262, for a full description of this method.

The latter option (\texttt{--polygon}) is a higher-level method to define any polygon (with any number of vertices) with floating point values. Please see the description of this option in Section 6.1.4 [Invoking Crop], page 263, for its syntax.

WCS coordinates

In WCS mode (\texttt{--mode=wcs}), the coordinates and width are interpreted using the World Coordinate System (WCS, that must accompany the dataset), not
pixel coordinates. You can optionally use \texttt{--widthinpix} for the width to be interpreted in pixels (even though the coordinates are in WCS). In WCS mode, Crop accepts multiple datasets as input. When the cropped region (defined by its center or vertices) overlaps with multiple of the input images/tiles, the overlapping regions will be taken from the respective input (they will be stitched when necessary for each output crop).

In this mode, the input images do not necessarily have to be the same size, they just need to have the same orientation and pixel resolution. Currently only orientation along the celestial coordinates is accepted, if your input has a different orientation you can use Warp’s \texttt{--align} option to align the image before cropping it (see Section 6.4 [Warp], page 328).

Each individual input image/tile can even be smaller than the final crop. In any case, any part of any of the input images which overlaps with the desired region will be used in the crop. Note that if there is an overlap in the input images/tiles, the pixels from the last input image read are going to be used for the overlap. Crop will not change pixel values, so it assumes your overlapping tiles were cutout from the same original image. There are multiple ways to define your cropped region as listed below.

Center of multiple crops (in a catalog)

Similar to catalog inputs in Image mode (above), except that the values along each dimension are assumed to have the same units as the dataset’s WCS information. For example, the central RA and Dec value for each crop will be read from the first and second calls to the \texttt{--coordcol} option. The width of the cropped box (in units of the WCS, or degrees in RA and Dec mode) must be specified with the \texttt{--width} option. You can optionally use \texttt{--widthinpix} for the value of \texttt{--width} to be interpreted in pixels.

Center of a single crop (on the command-line)

You can specify the center of only one crop box with the \texttt{--center} option. If it exists in the input images, it will be cropped similar to the catalog mode, see above also for \texttt{--width}.

Vertices of a single crop

The \texttt{--polygon} option is a high-level method to define any convex polygon (with any number of vertices). Please see the description of this option in Section 6.1.4 [Invoking Crop], page 263, for its syntax.

\textbf{CAUTION:} In WCS mode, the image has to be aligned with the celestial coordinates, such that the first FITS axis is parallel (opposite direction) to the Right Ascension (RA) and the second FITS axis is parallel to the declination. If these conditions aren’t met for an image, Crop will warn you and abort. You can use Warp’s \texttt{--align} option to align the input image with these coordinates, see Section 6.4 [Warp], page 328.
As a summary, if you don’t specify a catalog, you have to define the cropped region manually on the command-line. In any case the mode is mandatory for Crop to be able to interpret the values given as coordinates or widths.

6.1.2 Crop section syntax

When in image mode, one of the methods to crop only one rectangular section from the input image is to use the \texttt{--section} option. Crop has a powerful syntax to read the box parameters from a string of characters. If you leave certain parts of the string to be empty, Crop can fill them for you based on the input image sizes.

To define a box, you need the coordinates of two points: the first \((X_1, Y_1)\) and the last pixel \((X_2, Y_2)\) pixel positions in the image, or four integer numbers in total. The four coordinates can be specified with one string in this format: \('X_1:X_2,Y_1:Y_2'\). This string is given to the \texttt{--section} option. Therefore, the pixels along the first axis that are \(\geq X_1\) and \(\leq X_2\) will be included in the cropped image. The same goes for the second axis. Note that each different term will be read as an integer, not a float.

The reason it only accepts integers is that \texttt{--section} is a low-level option (which is also very fast!). For a higher-level way to specify region (any polygon, not just a box), please see the \texttt{--polygon} option in Section 6.1.4.1 [Crop options], page 264. Also note that in the FITS standard, pixel indexes along each axis start from unity(1) not zero(0).

You can omit any of the values and they will be filled automatically. The left hand side of the colon (:) will be filled with 1, and the right side with the image size. So, \(2:,2\) will include the full range of pixels along the second axis and only those with a first axis index larger than 2 in the first axis. If the colon is omitted for a dimension, then the full range is automatically used. So the same string is also equal to \(2:,2\) or \(2:,2\) or even \(2\). If you want such a case for the second axis, you should set it to: \(,2\).

If you specify a negative value, it will be seen as before the indexes of the image which are outside the image along the bottom or left sides when viewed in SAO DS9. In case you want to count from the top or right sides of the image, you can use an asterisk (*). When confronted with a *, Crop will replace it with the maximum length of the image in that dimension. So \(*-10:*+10,*-20:*+20\) will mean that the crop box will be 20 \(\times\) 40 pixels in size and only include the top corner of the input image with 3/4 of the image being covered by blank pixels, see Section 6.1.3 [Blank pixels], page 262.

If you feel more comfortable with space characters between the values, you can use as many space characters as you wish, just be careful to put your value in double quotes, for example \texttt{--section="5:200, 123:854"}. If you forget the quotes, anything after the first space will not be seen by \texttt{--section} and you will most probably get an error because the rest of your string will be read as a filename (which most probably doesn’t exist). See Section 4.1 [Command-line], page 152, for a description of how the command-line works.

6.1.3 Blank pixels

The cropped box can potentially include pixels that are beyond the image range. For example when a target in the input catalog was very near the edge of the input image. The parts of the cropped image that were not in the input image will be filled with the following two values depending on the data type of the image. In both cases, SAO DS9 will not color code those pixels.
• If the data type of the image is a floating point type (float or double), IEEE NaN (Not a number) will be used.
• For integer types, pixels out of the image will be filled with the value of the BLANK keyword in the cropped image header. The value assigned to it is the lowest value possible for that type, so you will probably never need it any way. Only for the unsigned character type (BITPIX=8 in the FITS header), the maximum value is used because it is unsigned, the smallest value is zero which is often meaningful.

You can ask for such blank regions to not be included in the output crop image using the --noblank option. In such cases, there is no guarantee that the image size of your outputs are what you asked for.

In some survey images, unfortunately they do not use the BLANK FITS keyword. Instead they just give all pixels outside of the survey area a value of zero. So by default, when dealing with float or double image types, any values that are 0.0 are also regarded as blank regions. This can be turned off with the --zeroisnotblank option.

6.1.4 Invoking Crop

Crop will crop a region from an image. If in WCS mode, it will also stitch parts from separate images in the input files. The executable name is astcrop with the following general template

```
$ astcrop [OPTION...] [ASCIIcatalog] ASTRdata ...
```

One line examples:

```
## Crop all objects in cat.txt from image.fits:
$ astcrop --catalog=cat.txt image.fits

## Crop all options in catalog (with RA,DEC) from all the files
## ending in '_drz.fits' in '/mnt/data/COSMOS/':
$ astcrop --mode=wcs --catalog=cat.txt /mnt/data/COSMOS/*_drz.fits

## Crop the outer 10 border pixels of the input image:
$ astcrop --section=10:*-10,10:*-10 --hdu=2 image.fits

## Crop region around RA and Dec of (189.16704, 62.218203):
$ astcrop --mode=wcs --center=189.16704,62.218203 goodsnorth.fits

## Same crop above, but coordinates given in sexagesimal (you can
## also use ':' between the sexagesimal components).
$ astcrop --mode=wcs --center=12h36m40.08,62d13m5.53 goodsnorth.fits

## Crop region around pixel coordinate (568.342, 2091.719):
$ astcrop --mode=img --center=568.342,2091.719 --width=201 image.fits
```

Crop has one mandatory argument which is the input image name(s), shown above with ASTRdata .... You can use shell expansions, for example * for this if you have lots of images in WCS mode. If the crop box centers are in a catalog, you can use the --catalog option. In other cases, you have to provide the single cropped output parameters must be given with command-line options. See Section 6.1.4.2 [Crop output], page 270, for how the
output file name(s) can be specified. For the full list of general options to all Gnuastro programs (including Crop), please see Section 4.1.2 [Common options], page 156.

Floating point numbers can be used to specify the crop region (except the --section option, see Section 6.1.2 [Crop section syntax], page 262). In such cases, the floating point values will be used to find the desired integer pixel indices based on the FITS standard. Hence, Crop ultimately doesn’t do any sub-pixel cropping (in other words, it doesn’t change pixel values). If you need such crops, you can use Section 6.4 [Warp], page 328, to first warp the image to the a new pixel grid, then crop from that. For example, let’s assume you want a crop from pixels 12.982 to 80.982 along the first dimension. You should first translate the image by $-0.482$ (note that the edge of a pixel is at integer multiples of 0.5). So you should run Warp with --translate=-0.482,0 and then crop the warped image with --section=13:81.

There are two ways to define the cropped region: with its center or its vertices. See Section 6.1.1 [Crop modes], page 259, for a full description. In the former case, Crop can check if the central region of the cropped image is indeed filled with data or is blank (see Section 6.1.3 [Blank pixels], page 262), and not produce any output when the center is blank, see the description under --checkcenter for more.

When in catalog mode, Crop will run in parallel unless you set --numthreads=1, see Section 4.4 [Multi-threaded operations], page 175. Note that when multiple outputs are created with threads, the outputs will not be created in the same order. This is because the threads are asynchronous and thus not started in order. This has no effect on each output, see Section 2.2.19 [Finding reddest clumps and visual inspection], page 68, for a tutorial on effectively using this feature.

6.1.4.1 Crop options

The options can be classified into the following contexts: Input, Output and operating mode options. Options that are common to all Gnuastro program are listed in Section 4.1.2 [Common options], page 156, and will not be repeated here.

When you are specifying the crop vertices yourself (through --section, or --polygon) on relatively small regions (depending on the resolution of your images) the outputs from image and WCS mode can be approximately equivalent. However, as the crop sizes get large, the curved nature of the WCS coordinates have to be considered. For example, when using --section, the right ascension of the bottom left and top left corners will not be equal. If you only want regions within a given right ascension, use --polygon in WCS mode.

Input image parameters:

--hstartwcs=INT

Specify the first keyword card (line number) to start finding the input image world coordinate system information. This is useful when certain header keywords of the input may cause bad conflicts with your crop (see an example described below). To get line numbers of the header keywords, you can pipe the fully printed header into cat -n like below:

```bash
$ astfits image.fits -h1 | cat -n
```

For example, distortions have only been present in WCSLIB from version 5.15 (released in mid 2016). Therefore some pipelines still apply their own specific
set of WCS keywords for distortions and put them into the image header along with those that WCSLIB does recognize. So now that WCSLIB recognizes most of the standard distortion parameters, they will get confused with the old ones and give wrong results. For example in the CANDELS-GOODS South images that were created before WCSLIB 5.15\(^1\).

The two --hstartwcs and --hendwcs are thus provided so when using older datasets, you can specify what region in the FITS headers you want to use to read the WCS keywords. Note that this is only relevant for reading the WCS information, basic data information like the image size are read separately. These two options will only be considered when the value to --hendwcs is larger than that of --hstartwcs. So if they are equal or --hstartwcs is larger than --hendwcs, then all the input keywords will be parsed to get the WCS information of the image.

--hendwcs=INT
Specify the last keyword card to read for specifying the image world coordinate system on the input images. See --hstartwcs

Crop box parameters:

-c FLT[,FLT[,...]]
--center=FLT[,FLT[,...]]
The central position of the crop in the input image. The positions along each dimension must be separated by a comma (,) and fractions are also acceptable. The comma-separated values can either be in degrees (a single number), or sexagesimal (\_h\_m\_ for RA, \_d\_m\_ for Dec, or \_:\_:\_ for both).

The number of values given to this option must be the same as the dimensions of the input dataset. The width of the crop should be set with --width. The units of the coordinates are read based on the value to the --mode option, see below.

-w FLT[,FLT[,...]]
--width=FLT[,FLT[,...]]
Width of the cropped region about coordinate given to --center. If in WCS mode, value(s) given to this option will be read in the same units as the dataset’s WCS information along this dimension (unless --widthinpix is given). This option may take either a single value (to be used for all dimensions: --width=10 in image-mode will crop a 10 \times 10 pixel image) or multiple values (a specific value for each dimension: --width=10,20 in image-mode will crop a 10 \times 20 pixel image).

The --width option also accepts fractions. For example if you want the width of your crop to be 3 by 5 arcseconds along RA and Dec respectively and you are in wcs-mode, you can use: --width=3/3600,5/3600.

The final output will have an odd number of pixels to allow easy identification of the pixel which keeps your requested coordinate (from --center or --catalog). If you want an even sided crop, you can run Crop afterwards with

\(^1\) https://archive.stsci.edu/pub/hlsp/candels/goods-s/gs-tot/v1.0/
--section=':*-1,:*-1' or --section=2:,2: (depending on which side you don’t need), see Section 6.1.2 [Crop section syntax], page 262.

The basic reason for making an odd-sided crop is that your given central coordinate will ultimately fall within a discrete pixel in the image (defined by the FITS standard). When the crop has an odd number of pixels in each dimension, that pixel can be very well defined as the “central” pixel of the crop, making it unambiguously easy to identify. However, for an even-sided crop, it will be very hard to identify the central pixel (it can be on any of the four pixels adjacent to the central point of the image!).

-X
--widthinpix
In WCS mode, interpret the value to --width as number of pixels, not the WCS units like degrees. This is useful when you want a fixed crop size in pixels, even though your center coordinates are in WCS (for example RA and Dec).

-l STR
-l FLT:FLT,...
--polygon=STR
--polygon=FLT,FLT:FLT,...
Polygon vertice coordinates (when value is in FLT,FLT:FLT,FLT:... format) or the filename of a SAO DS9 region file (when the value has no , or : characters). Each vertice can either be in degrees (a single floating point number) or sexagesimal (in formats of '_h_m_' for RA and '_d_m_' for Dec, or simply '_:_:_' for either of them).

The vertices are used to define the polygon: in the same order given to this option. When the vertices are not necessarily ordered in the proper order (for example one vertice in a square comes after its diagonal opposite), you can add the --polygonsort option which will attempt to sort the vertices before cropping. Note that for concave polygons, sorting is not recommended because there is no unique solution, for more, see the description under --polygonsort.

This option can be used both in the image and WCS modes, see Section 6.1.1 [Crop modes], page 259. If a SAO DS9 region file is used, the coordinate mode of Crop will be determined by the contents of the file and any value given to --mode is ignored. The cropped image will be the size of the rectangular region that completely encompasses the polygon. By default all the pixels that are outside of the polygon will be set as blank values (see Section 6.1.3 [Blank pixels], page 262). However, if --polygonout is called all pixels internal to the vertices will be set to blank. In WCS-mode, you may provide many FITS images/tiles: Crop will stitch them to produce this cropped region, then apply the polygon.

The syntax for the polygon vertices is similar to, and simpler than, that for --section. In short, the dimensions of each coordinate are separated by a comma (,) and each vertex is separated by a colon (;). You can define as many vertices as you like. If you would like to use space characters between the dimensions and vertices to make them more human-readable, then you have to put the value to this option in double quotation marks.
For example, let’s assume you want to work on the deepest part of the WFC3/IR images of Hubble Space Telescope eXtreme Deep Field (HST-XDF). According to the web page (https://archive.stsci.edu/prepds/xdf/)\(^2\) the deepest part is contained within the coordinates:

\[
(53.187414,-27.779152), (53.159507,-27.759633), (53.134517,-27.787144), (53.161906,-27.807208)
\]

They have provided mask images with only these pixels in the WFC3/IR images, but what if you also need to work on the same region in the full resolution ACS images? Also what if you want to use the CANDELS data for the shallow region? Running Crop with --polygon will easily pull out this region of the image for you, irrespective of the resolution. If you have set the operating mode to WCS mode in your nearest configuration file (see Section 4.2 [Configuration files], page 169), there is no need to call --mode=wcs on the command line.

```bash
$ astcrop --mode=wcs desired-filter-image(s).fits \\
   --polygon="53.187414,-27.779152 : 53.159507,-27.759633 : \\
                  53.134517,-27.787144 : 53.161906,-27.807208"
```

More generally, you have an image and want to define the polygon yourself (it isn’t already published like the example above). As the number of vertices increases, checking the vertex coordinates on a FITS viewer (for example SAO DS9) and typing them in, one by one, can be very tedious and prone to typo errors. In such cases, you can make a polygon “region” in DS9 and using your mouse, easily define (and visually see) it. Given that SAO DS9 has a graphic user interface (GUI), if you don’t have the polygon vertices before-hand, it is much more easier build your polygon there and pass it onto Crop through the region file.

You can take the following steps to make an SAO DS9 region file containing your polygon. Open your desired FITS image with SAO DS9 and activate its “region” mode with Edit→Region. Then define the region as a polygon with Region→Shape→Polygon. Click on the approximate center of the region you want and a small square will appear. By clicking on the vertices of the square you can shrink or expand it, clicking and dragging anywhere on the edges will enable you to define a new vertex. After the region has been nicely defined, save it as a file with Region→“Save Regions”. You can then select the name and address of the output file, keep the format as REG (*.reg) and press the “OK” button. In the next window, keep format as “ds9” and “Coordinate System” as “fk5” for RA and Dec (or “Image” for pixel coordinates). A plain text file is now created (let’s call it ds9.reg) which you can pass onto Crop with --polygon=ds9.reg.

For the expected format of the region file, see the description of gal_ds9_reg_read_polygon in Section 11.3.31 [SAO DS9 library (ds9.h)], page 661. However, since SAO DS9 makes this file for you, you don’t usually need to worry about its internal format unless something un-expected happens and you find a bug.

\(^2\) https://archive.stsci.edu/prepds/xdf/
--polygonout

Keep all the regions outside the polygon and mask the inner ones with blank pixels (see Section 6.1.3 [Blank pixels], page 262). This is practically the inverse of the default mode of treating polygons. Note that this option only works when you have only provided one input image. If multiple images are given (in WCS mode), then the full area covered by all the images has to be shown and the polygon excluded. This can lead to a very large area if large surveys like COSMOS are used. So Crop will abort and notify you. In such cases, it is best to crop out the larger region you want, then mask the smaller region with this option.

--polygonsort

Sort the given set of vertices to the --polygon option. For a concave polygon it will sort the vertices correctly, however for a convex polygon it there is no unique sorting, so be careful because the crop may not be what you expected.

Polygons come in two classes: convex and concave (or generally, non-convex!), see below for a demonstration. Convex polygons are those where all inner angles are less than 180 degrees. By contrast, a concave polygon is one where an inner angle may be more than 180 degrees.

<table>
<thead>
<tr>
<th>Concave Polygon</th>
<th>Convex Polygon</th>
</tr>
</thead>
<tbody>
<tr>
<td>D -------- C</td>
<td>D------------- C</td>
</tr>
<tr>
<td>\</td>
<td>\</td>
</tr>
<tr>
<td>\E</td>
<td>\</td>
</tr>
<tr>
<td>/</td>
<td>\</td>
</tr>
<tr>
<td>A--------B</td>
<td>A ----------- B</td>
</tr>
</tbody>
</table>

-s STR

--section=STR

Section of the input image which you want to be cropped. See Section 6.1.2 [Crop section syntax], page 262, for a complete explanation on the syntax required for this input.

-C FITS/TXT

--catalog=FITS/TXT

File name of catalog for making multiple crops from the input images/cubes. The catalog can be in any of Gnuastro’s recognized Section 4.7.1 [Recognized table formats], page 184. The columns containing the coordinates for the crop centers can be specified with the --coordcol option (using column names or numbers, see Section 4.7.3 [Selecting table columns], page 188). The catalog can also contain the name of each crop, you can specify the column containing the name with the --namecol.

--cathdu=STR/INT

The HDU (extension) containing the catalog (if the file given to --catalog is a FITS file). This can either be the HDU name (if it has one) or number (counting from 0). By default (if this option is not given), the second HDU will be used (equivalent to --cathdu=1. For more on how to specify the HDU, see
the explanation of the --hdu option in Section 4.1.2.1 [Input/Output options], page 157.

-x STR/INT
--coordcol=STR/INT
The column in a catalog to read as a coordinate. The value can be either the column number (starting from 1), or a match/search in the table meta-data, see Section 4.7.3 [Selecting table columns], page 188. This option must be called multiple times, depending on the number of dimensions in the input dataset. If it is called more than necessary, the extra columns (later calls to this option on the command-line or configuration files) will be ignored, see Section 4.2.2 [Configuration file precedence], page 170.

-n STR/INT
--namecol=STR/INT
Column selection of crop file name. The value can be either the column number (starting from 1), or a match/search in the table meta-data, see Section 4.7.3 [Selecting table columns], page 188. This option can be used both in Image and WCS modes, and not a mandatory. When a column is given to this option, the final crop base file name will be taken from the contents of this column. The directory will be determined by the --output option (current directory if not given) and the value to --suffix will be appended. When this column isn’t given, the row number will be used instead.

Output options:

-c FLT/INT
--checkcenter=FLT/INT
Square box width of region in the center of the image to check for blank values. If any of the pixels in this central region of a crop (defined by its center) are blank, then it will not be stored in an output file. If the value to this option is zero, no checking is done. This check is only applied when the cropped region(s) are defined by their center (not by the vertices, see Section 6.1.1 [Crop modes], page 259).

The units of the value are interpreted based on the --mode value (in WCS or pixel units). The ultimate checked region size (in pixels) will be an odd integer around the center (converted from WCS, or when an even number of pixels are given to this option). In WCS mode, the value can be given as fractions, for example if the WCS units are in degrees, 0.1/3600 will correspond to a check size of 0.1 arcseconds.

Because survey regions don’t often have a clean square or rectangle shape, some of the pixels on the sides of the survey FITS image don’t commonly have any data and are blank (see Section 6.1.3 [Blank pixels], page 262). So when the catalog was not generated from the input image, it often happens that the image does not have data over some of the points.

When the given center of a crop falls in such regions or outside the dataset, and this option has a non-zero value, no crop will be created. Therefore with this option, you can specify a width of a small box (3 pixels is often good enough)
around the central pixel of the cropped image. You can check which crops were created and which weren’t from the command-line (if --quiet was not called, see Section 4.1.2.3 [Operating mode options], page 162), or in Crop’s log file (see Section 6.1.4.2 [Crop output], page 270).

-p STR
--suffix=STR
The suffix (or post-fix) of the output files for when you want all the cropped images to have a special ending. One case where this might be helpful is when besides the science images, you want the weight images (or exposure maps, which are also distributed with survey images) of the cropped regions too. So in one run, you can set the input images to the science images and --suffix=_s.fits. In the next run you can set the weight images as input and --suffix=_w.fits.

--primaryimghdu
Write the output into the primary (0-th) HDU/extension of the output. By default, like all Gnuastro’s default outputs, no data is written in the primary extension because the FITS standard suggests keeping that extension free of data and only for meta data.

-b
--noblank
Pixels outside of the input image that are in the crop box will not be used. By default they are filled with blank values (depending on type), see Section 6.1.3 [Blank pixels], page 262. This option only applies only in Image mode, see Section 6.1.1 [Crop modes], page 259.

-z
--zeroisnotblank
In float or double images, it is common to give the value of zero to blank pixels. If the input image type is one of these two types, such pixels will also be considered as blank. You can disable this behavior with this option, see Section 6.1.3 [Blank pixels], page 262.

Operating mode options:

-0 STR
--mode=STR
Operate in Image mode or WCS mode when the input coordinates can be both image or WCS. The value must either be img or wcs, see Section 6.1.1 [Crop modes], page 259, for a full description.

6.1.4.2 Crop output

The string given to --output option will be interpreted depending on how many crops were requested, see Section 6.1.1 [Crop modes], page 259:

- When a catalog is given, the value of the --output (see Section 4.1.2 [Common options], page 156) will be read as the directory to store the output cropped images. Hence if it doesn’t already exist, Crop will abort with an “No such file or directory” error.
The crop file names will consist of two parts: a variable part (the row number of each target starting from 1) along with a fixed string which you can set with the \texttt{--suffix} option. Optionally, you may also use the \texttt{--namecol} option to define a column in the input catalog to use as the file name instead of numbers.

- When only one crop is desired, the value to \texttt{--output} will be read as a file name. If no output is specified or if it is a directory, the output file name will follow the automatic output names of Gnuastro, see Section 4.9 [Automatic output], page 191: The string given to \texttt{--suffix} will be replaced with the \texttt{.fits} suffix of the input.

By default, as suggested by the FITS standard and implemented in all Gnuastro programs, the first/primary extension of the output files will only contain meta data. The cropped images/cubes will be written into the 2nd HDU of their respective FITS file (which is actually counted as 1 because HDU counting starts from 0). However, if you want the cropped data to be written into the primary (0-th) HDU, run Crop with the \texttt{--primaryimghdu} option.

The header of each output cropped image will contain the names of the input image(s) it was cut from. If a name is longer than the 70 character space that the FITS standard allows for header keyword values, the name will be cut into several keywords from the nearest slash (/). The keywords have the following format: \texttt{ICFn_m} (for Crop File). Where \texttt{n} is the number of the image used in this crop and \texttt{m} is the part of the name (it can be broken into multiple keywords). Following the name is another keyword named \texttt{ICFnPIX} which shows the pixel range from that input image in the same syntax as Section 6.1.2 [Crop section syntax], page 262. So this string can be directly given to the \texttt{--section} option later.

Once done, a log file can be created in the current directory with the \texttt{--log} option. This file will have three columns and the same number of rows as the number of cropped images. There are also comments on the top of the log file explaining basic information about the run and descriptions for the columns. A short description of the columns is also given below:

1. The cropped image file name for that row.
2. The number of input images that were used to create that image.
3. A 0 if the central few pixels (value to the \texttt{--checkcenter} option) are blank and 1 if they aren’t. When the crop was not defined by its center (see Section 6.1.1 [Crop modes], page 259), or \texttt{--checkcenter} was given a value of 0 (see Section 6.1.4 [Invoking Crop], page 263), the center will not be checked and this column will be given a value of -1.

### 6.1.4.3 Crop known issues

When running Crop, you may encounter strange errors and bugs. In these cases, please report a bug and we will try to fix it as soon as possible, see Section 1.8 [Report a bug], page 11. However, some things are beyond our control, or may take too long to fix directly. In this section we list such known issues that may occur in known cases and suggest the hack (or work-around) to fix the problem:

**Crash with ‘killed’ when cropping catalog from \texttt{.fits.gz}**

This happens because CFISTIO (that reads and writes FITS files) will internally decompress the file in a temporary place (possibly in the RAM), then start
reading from it. On the other hand, by default when given a catalog (with many crops) and not specifying --numthreads, Crop will use the maximum number of threads available on your system to do each crop faster. On an normal (not compressed) file, parallel access will not cause a problem, however, when attempting parallel access with the maximum number of threads on a compressed file, CFITSIO crashes with Killed. Therefore the following solutions can be used to fix this crash:

- Decrease the number of threads (at the minimum, set --numthreads=1). Since this solution doesn’t attempt to change any of your previous Crop command components or doesn’t change your local file structure, it is the preferred way.
- Decompress the file (with the command below) and feed the .fits file into Crop without changing the number of threads.

```
$ gunzip -k image.fits.gz
```

### 6.2 Arithmetic

It is commonly necessary to do operations on some or all of the elements of a dataset independently (pixels in an image). For example, in the reduction of raw data it is necessary to subtract the Sky value (Section 7.1.4 [Sky value], page 344) from each image image. Later (once the images as warped into a single grid using Warp for example, see Section 6.4 [Warp], page 328), the images are co-added (the output pixel grid is the average of the pixels of the individual input images). Arithmetic is Gnuastro’s program for such operations on your datasets directly from the command-line. It currently uses the reverse polish or post-fix notation, see Section 6.2.1 [Reverse polish notation], page 272, and will work on the native data types of the input images/data to reduce CPU and RAM resources, see Section 4.5 [Numeric data types], page 178. For more information on how to run Arithmetic, please see Section 6.2.4 [Invoking Arithmetic], page 302.

#### 6.2.1 Reverse polish notation

The most common notation for arithmetic operations is the infix notation (https://en.wikipedia.org/wiki/Infix_notation) where the operator goes between the two operands, for example $4 + 5$. The infix notation is the preferred way in most programming languages which come with scripting features for large programs. This is because the infix notation requires a way to define precedence when more than one operator is involved.

For example consider the statement $5 + 6 / 2$. Should 6 first be divided by 2, then added by 5? Or should 5 first be added with 6, then divided by 2? Therefore we need parenthesis to show precedence: $(5 + 6)/2$ or $(5+6)/2$. Furthermore, if you need to leave a value for later processing, you will need to define a variable for it; for example $a=(5+6)/2$.

Gnuastro provides libraries where you can also use infix notation in C or C++ programs. However, Gnuastro’s programs are primarily designed to be run on the command-line and the level of complexity that infix notation requires can be annoying/confusing to write on the command-line (where they can get confused with the shell’s parenthesis or variable definitions). Therefore Gnuastro’s Arithmetic and Table (when doing column arithmetic) programs use the post-fix notation, also known as reverse polish notation (https://en.
wikipedia.org/wiki/Reverse_Polish_notation). For example, instead of writing $5+6$, we write $5\ 6\ +$.

The Wikipedia article on the reverse polish notation provides some excellent explanation on this notation but here we will give a short summary here for self-sufficiency. In short, in the reverse polish notation, the operator is placed after the operands. As we will see below this removes the need to define parenthesis and lets you use previous values without needing to define a variable. In the future we do plan to also optionally allow infix notation when arithmetic operations on datasets are desired, but due to time constraints on the developers we can’t do it immediately.

To easily understand how the reverse polish notation works, you can think of each operand (5 and 6 in the example above) as a node in a “last-in-first-out” stack. One such stack in daily life is a stack of dishes in the kitchen: you put a clean dish, on the top of a stack of dishes when its ready for later usage. Later, when you need a dish, you pick the top one (hence the “last” dish placed “in” the stack is the “first” dish that comes “out” when necessary).

Each operator will need a certain number of operands (in the example above, the + operator needs two operands: 5 and 6). In the kitchen metaphor, an operator can be an oven. Every time an operator is confronted, the operator takes (or “pops”) the number of operands it needs from the top of the stack (so they don’t exist in the stack any more), does its operation, and places (or “pushes”) the result back on top of the stack. So if you want the average of 5 and 6, you would write: $5\ 6\ +\ 2\ /$. The operations that are done are:

1. 5 is an operand, so Arithmetic pushes it to the top of the stack (which is initially empty). In the kitchen metaphor, you can visualize this as taking a new dish from the cabinet, putting the number 5 inside of the dish, and putting the dish on top of the (empty) cooking table in front of you. You now have a stack of one dish on the table in front of you.

2. 6 is also an operand, so it is pushed to the top of the stack. Like before, you can visualize this as taking a new dish from the cabinet, putting the number 6 in it and placing it on top of the previous dish. You now have a stack of two dishes on the table in front of you.

3. + is a binary operator, so it will pop the top two elements of the stack out of it, and perform addition on them (the order is 5 + 6 in the example above). The result is 11 which is pushed to the top of the stack.

To visualize this, you can think of the + operator as an oven with a place for two dishes. You pick up the top-most dish (that has the number 6 in it) and put it in the oven. The top dish is now the one that has the number 5. You also pick it up and put it in the oven, and close the oven door. When the oven has finished its cooking, it produces a single output (in one dish, with the number 11 inside of it). You take that output dish and put it back on the table. You now have a stack of one dish on the table in front of you.

4. 2 is an operand so push it onto the top of the stack. In the kitchen metaphor, you again go to the cabinet, pick up a dish and put the number 2 inside of it and put the dish over the previous dish (that has the number 11). You now have a stack of two dishes on the table in front of you.

---

5. / (division) is a binary operator, so pull out the top two elements of the stack (top-most is 2, then 11) and divide the second one by the first. In the kitchen metaphor, the / operator can be visualized as a microwave that takes two dishes. But unlike the oven (+ operator) before, the order of inputs matters (they are on top of each other: with the top dish holder being the nominator and the bottom one being the denominator). Again, you look to your stack of dishes on the table.

You pick up the top one (with value 2 inside of it) and put it in the microwave’s bottom (denominator) dish holder. Then you go back to your stack of dishes on the table and pick up the top dish (with value 11 inside of it) and put that in the top (nominator) dish holder. The microwave will do its work and when its finished, returns a new dish with the single value 5.5 inside of it. You pick up the dish from the microwave and place it back on the table.

6. There are no more operands or operators, so simply return the remaining operand in the output. In the kitchen metaphor, you see that your recipe has no more steps, so you just pick up the remaining dish and take it to the dining room to enjoy a good dinner.

In the Arithmetic program, the operands can be FITS images of any dimensionality, or numbers (see Section 6.2.4 [Invoking Arithmetic], page 302). In Table’s column arithmetic, they can be any column in the table (a series of numbers in an array) or a single number (see Section 5.3.1 [Column arithmetic], page 230).

With this notation, very complicated procedures can be created without the need for parenthesis or worrying about precedence. Even functions which take an arbitrary number of arguments can be defined in this notation. This is a very powerful notation and is used in languages like Postscript which produces PDF files when compiled.

### 6.2.2 Integer benefits and pitfalls

Integers are the simplest numerical data types (Section 4.5 [Numeric data types], page 178). Because of this, their storage space is much less, and their processing is much faster than floating point types. You can confirm this on your computer with the series of commands below. You will make four 5000 by 5000 pixel images filled with random values. Two of them will be saved as signed 8-bit integers, and two with 64-bit floating point types. The last command prints the size of the created images.

```
$ astarithmetic 5000 5000 2 makenew 5 mknoise-sigma int8 -oint-1.fits
$ astarithmetic 5000 5000 2 makenew 5 mknoise-sigma int8 -oint-2.fits
$ astarithmetic 5000 5000 2 makenew 5 mknoise-sigma float64 -oflt-1.fits
$ astarithmetic 5000 5000 2 makenew 5 mknoise-sigma float64 -oflt-2.fits
$ ls -lh int-*.fits flt-*.fits
```

The 8-bit integer images are only 24MB, while the 64-bit floating point images are 191 MB! Besides helping in storage (on your disk, or in RAM, while the program is running), the small size of these files also helps in faster reading of the inputs. Furthermore, CPUs can process integer operations much faster than floating points. In the integers, the ones with a smaller width (number of bits) can be processed much faster. You can see this with

---

4 See the EPS and PDF part of Section 5.2.1 [Recognized file formats], page 211, for a little more on the Postscript language.
the two commands below where you will add the integer images with each other and the floats with each other:

```bash
$ astarithmetic flt-1.fits flt-2.fits + -o flt-sum.fits -g1
$ astarithmetic int-1.fits int-2.fits + -o int-sum.fits -g1
```

Have a look at the running time of the two commands above (that is printed on their last line). On the system that this paragraph was written on, the floating point and integer image sums were respectively done in 0.481 and 0.089 seconds (the integer operation was almost 5 times faster!).

| If your data doesn’t have decimal points, use integer types: | Integer types are much faster and can take much less space in your storage or RAM (while the program is running). |
| Select the smallest width that can host the range/precision of values: | For example, if the largest possible value in your dataset is 1000 and all numbers are integers, store it as a 16-bit integer. Also, if you know the values can never become negative, store it as an unsigned 16-bit integer. For floating point types, if you know you won’t need a precision of more than 6 significant digits, use the 32-bit floating point type. For more on the range (for integers) and precision (for floats), see Section 4.5 [Numeric data types], page 178. |

There is a price to be paid for this improved efficiency in integers: your wisdom! If you have not selected your types wisely, strange situations may happen. For example try the command below:

```bash
$ astarithmetic 125 10 +
```

You expect the output to be 135, but it will be \(-121\)! The reason is that when Arithmetic (or column-arithmetic in Table) confronts a number on the command-line, it use the principles above to select the most efficient type for each number. Both 125 and 10 can safely fit within a signed, 8-bit integer type, so arithmetic will store both as an 8-bit integer. However, the sum (135) is larger than the maximum possible value of an 8-bit signed integer (127). Therefore an integer overflow will occur, and the bits will be over-written. As a result, the value will be \(135 - 128 = 7\) more than the minimum value of this type (\(-128\)), which is \(-128 + 7 = -121\).

When you know situations like this may occur, you can simply use Section 6.2.3.12 [Numerical type conversion operators], page 296, to set just one of the inputs to a wider data type (the smallest, wider type to avoid wasting resources). In the example above, this would be `uint16`:

```bash
$ astarithmetic 125 uint16 10 +
```

The reason this worked is that 125 is now converted into an unsigned 16-bit integer before the `+` operator. Since this is larger than an 8-bit integer, the C programming language’s automatic type conversion will treat both as the wider type and store the result of the binary operation (`+`) in that type.

For such a basic operation like the command above, a faster hack would be any of the two commands below (which are equivalent). This is because `125.0` or `125.` are interpreted as floating-point types and they don’t suffer from such issues (converting only on one input is enough):
For this particular command, the fix above will be as fast as the `uint16` solution. This is because there are only two numbers, and the overhead of Arithmetic (reading configuration files, and etc) dominates the running time. However, for large datasets, the `uint16` solution will be faster (as you saw above), Arithmetic will consume less RAM while running, and the output will consume less storage in your system (all major benefits)!

It is possible to do internal checks in Gnuastro and catch integer overflows and correct them internally. However, we haven’t opted for this solution because all those checks will consume significant resources and slow down the program (especially with large datasets where RAM, storage and running time become important). To be optimal, we therefore trust that you (the wise Gnuastro user!) make the appropriate type conversion in your commands where necessary (recall that the operators are available in Section 6.2.3.12 [Numerical type conversion operators], page 296).

### 6.2.3 Arithmetic operators

In this section, list of recognized operators in Arithmetic (and the Table program’s Section 5.3.1 [Column arithmetic], page 230) and discussed in detail with examples. As mentioned before, to be able to easily do complex operations on the command-line, the Reverse Polish Notation is used (where you write ‘4 5 +’ instead of ‘4 + 5’), if you aren’t already familiar with it, before continuing, please see Section 6.2.1 [Reverse polish notation], page 272.

The operands to all operators can be a data array (for example a FITS image or data cube) or a number, the output will be an array or number according to the inputs. For example a number multiplied by an array will produce an array. The numerical data type of the output of each operator is described within it.

**Blank pixels in Arithmetic:** Blank pixels in the image (see Section 6.1.3 [Blank pixels], page 262) will be stored based on the data type. When the input is floating point type, blank values are NaN. One aspect of NaN values is that by definition they will fail on *any* comparison. Hence both equal and not-equal operators will fail when both their operands are NaN! Therefore, the only way to guarantee selection of blank pixels is through the `isblank` operator explained above.

One way you can exploit this property of the NaN value to your advantage is when you want a fully zero-valued image (even over the blank pixels) based on an already existing image (with same size and world coordinate system settings). The following command will produce this for you:

```bash
$ astarithmetic input.fits nan eq --output=all-zeros.fits
```

Note that on the command-line you can write NaN in any case (for example `NaN`, or `NAN` are also acceptable). Reading NaN as a floating point number in Gnuastro isn’t case-sensitive.
6.2.3.1 Basic mathematical operators

These are some of the most common operations you will be doing on your data and include, so no further explanation is necessary. If you are new to Gnuastro, just read the description of each carefully.

+ Addition, so “4 5 +” is equivalent to 4 + 5. For example in the command below, the value 20000 is added to each pixel’s value in image.fits:

$ astarithmetic 20000 image.fits +

You can also use this operator is to sum the values of one pixel in two images (which have to be the same size). For example in the commands below (which are identical, see paragraph after the commands), each pixel of sum.fits is the sum of the same pixel’s values in a.fits and b.fits.

$ astarithmetic a.fits b.fits + -h1 -h1 --output=sum.fits

$ astarithmetic a.fits b.fits + -g1 --output=sum.fits

The HDU/extension has to be specified for each image with -h. However, if the HDUs are the same in all inputs, you can use -g to only specify the HDU once.

If you need to add more than one dataset, one way is to use this operator multiple times, for example see the two commands below that are identical in the Reverse Polish Notation (Section 6.2.1 [Reverse polish notation], page 272):

$ astarithmetic a.fits b.fits + c.fits + -osum.fits

$ astarithmetic a.fits b.fits c.fits ++ -osum.fits

However, this can get annoying/buggy if you have more than three or four images, in that case, a better way to sum data is to use the sum operator (which also ignores blank pixels), that is discussed below.

- Subtraction, so “4 5 −” is equivalent to 4 − 5. Usage of this operator is similar to + operator, for example:

$ astarithmetic 20000 image.fits -

$ astarithmetic a.fits b.fits - -g1 --output=sub.fits

x Multiplication, so “4 5 x” is equivalent to 4 \times 5. For example in the command below, the value of each output pixel is 5 times its value in image.fits:

$ astarithmetic image.fits 5 x

And you can multiply the value of each pixel in two images, like this:

$ astarithmetic a.fits a.fits x -g1 --output=multip.fits

/ Division, so “4 5 /” is equivalent to 4/5. Like the multiplication, for example:

$ astarithmetic image.fits 5 -h1 /

$ astarithmetic a.fits b.fits / -g1 --output=div.fits

% Modulo (remainder), so “3 2 %” will return 1. Note that the modulo operator only works on integer types (see Section 4.5 [Numeric data types], page 178). This operator is therefore not defined for most processed astronomical astronomical images that have floating-point value. However it is useful in labeled images, for example Section 7.3.1.3 [Segment output], page 390). In such cases, each pixel is the integer label of the object it is associated with hence with the example command below, we can change the labels to only be between 1 and 4
Chapter 6: Data manipulation

and decrease all objects on the image to 4/5th (all objects with a label that is a multiple of 5 will be set to 0).

$ astrarithmetic label.fits 5 1$

**abs** Absolute value of first operand, so “4 abs” is equivalent to |4|. For example the output of the command bellow will not have any negative pixels (all negative pixels will be multiplied by −1 to become positive)

$ astrarithmetic image.fits abs$

**pow** First operand to the power of the second, so “4.3 5 pow” is equivalent to $4^{3.5}$. For example with the command below all pixels will be squared

$ astrarithmetic image.fits 2 pow$

**sqrt** The square root of the first operand, so “5 sqrt” is equivalent to $\sqrt{5}$. Since the square root is only defined for positive values, any negative-valued pixel will become NaN (blank). The output will have a floating point type, but its precision is determined from the input: if the input is a 64-bit floating point, the output will also be 64-bit. Otherwise, the output will be 32-bit floating point (see Section 4.5 [Numeric data types], page 178, for the respective precision). Therefore if you require 64-bit precision in estimating the square root, convert the input to 64-bit floating point first, for example with 5 float64 sqrt. For example each pixel of the output of the command below will be the square root of that pixel in the input.

$ astrarithmetic image.fits sqrt$

If you just want to scale an image with negative values using this operator (for better visual inspection, and the actual values don’t matter for you), you can subtract the image from its minimum value, then take its square root:

$ astrarithmetic image.fits image.fits minvalue - sqrt -g1$

Alternatively, to avoid reading the image into memory two times, you can use the set- operator to read it into the variable i and use i two times to speed up the operation (described below):

$ astrarithmetic image.fits set-i i i minvalue - sqrt$

**log** Natural logarithm of first operand, so “4 log” is equivalent to $ln(4)$. Negative pixels will become NaN, and the output type is determined from the input, see the explanation under sqrt for more on these features. For example the command below will take the natural logarithm of every pixel in the input.

$ astrarithmetic image.fits log --output=log.fits$

**log10** Base-10 logarithm of first popped operand, so “4 log10” is equivalent to $log_{10}(4)$. Negative pixels will become NaN, and the output type is determined from the input, see the explanation under sqrt for more on these features. For example the command below will take the base-10 logarithm of every pixel in the input.

$ astrarithmetic image.fits log10$
6.2.3.2 Trigonometric and hyperbolic operators

All the trigonometric and hyperbolic functions are described here. One good thing with these operators is that they take inputs and outputs in degrees (which we usually need as input or output), not radians (like most other programs/libraries).

- **sin**
  - Basic trigonometric functions. They take one operand, in units of degrees.
- **cos**
- **tan**
  - Inverse trigonometric functions. They take one operand and the returned values are in units of degrees.
- **atan**
  - Inverse tangent (output in units of degrees) that uses the signs of the input coordinates to distinguish between the quadrants. This operator therefore needs two operands: the first popped operand is assumed to be the X axis position of the point, and the second popped operand is its Y axis coordinate.

For example see the commands below. To be more clear, we are using Table's Section 5.3.1 [Column arithmetic], page 230, which uses exactly the same internal library function as the Arithmetic program for images. We are showing the results for four points in the four quadrants of the 2D space (if you want to try running them, you don’t need to type/copy the parts after #). The first point (2,2) is in the first quadrant, therefore the returned angle is 45 degrees. But the second, third and fourth points are in the quadrants of the same order, and the returned angles reflect the quadrant.

```
$ echo " 2 2" | asttable -c'arith $2 $1 atan2' # --> 45
$ echo " 2 -2" | asttable -c'arith $2 $1 atan2' # --> -45
$ echo "-2 -2" | asttable -c'arith $2 $1 atan2' # --> 135
$ echo "-2 2" | asttable -c'arith $2 $1 atan2' # --> -135
```

However, if you simply use the classic arc-tangent operator (**atan**) for the same points, the result will only be in two quadrants as you see below:

```
$ echo " 2 2" | asttable -c'arith $2 $1 / atan' # --> 45
$ echo " 2 -2" | asttable -c'arith $2 $1 / atan' # --> -45
$ echo "-2 -2" | asttable -c'arith $2 $1 / atan' # --> 45
$ echo "-2 2" | asttable -c'arith $2 $1 / atan' # --> -45
```

- **sinh**
- **cosh**
- **tanh**
  - Hyperbolic sine, cosine, and tangent. These operators take a single operand.
- **asinh**
- **acosh**
- **atanh**
  - Inverse Hyperbolic sine, cosine, and tangent. These operators take a single operand.

6.2.3.3 Unit conversion operators

It often happens that you have data in one unit (for example magnitudes to measure the brightness of a galaxy), but would like to convert it into another (for example electron
counts on your CCD). While the equations for the unit conversions can be easily found on the internet, the operators in this section are designed to simplify the process and let you do it easily.

**counts-to-mag**
Convert counts (usually CCD outputs) to magnitudes using the given zero point. The zero point is the first popped operand and the count image or value is the second popped operand.

For example assume you have measured the standard deviation of the noise in an image to be 0.1 counts, and the image’s zero point is 22.5 and you want to measure the per-pixel surface brightness limit of the dataset. To apply this operator on an image, simply replace 0.1 with the image name, as described below.

```
$ astarithmetic 0.1 22.5 counts-to-mag --quiet
```

Of course, you can also convert every pixel in an image (or table column in Table’s Section 5.3.1 [Column arithmetic], page 230) with this operator if you replace the second popped operand with an image/column name. For an example of applying this operator on an image, see the description of surface brightness in Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394, where we’ll convert an image’s pixel values to surface brightness.

**mag-to-counts**
Convert magnitudes to counts (usually CCD outputs) using the given zero point. The zero point is the first popped operand and the magnitude value is the second. For example if an object has a magnitude of 20, you can estimate the counts corresponding to it (when the image has a zero point of 24.8) with this command: Note that because the output is a single number, we are using --quiet to avoid printing extra information.

```
$ astarithmetic 20 24.8 mag-to-counts --quiet
```

**counts-to-jy**
Convert counts (usually CCD outputs) to Janskys through an AB-magnitude based zero point. The top-popped operand is assumed to be the AB-magnitude zero point and the second-popped operand is assumed to be a dataset in units of counts (an image in Arithmetic, and a column in Table’s Section 5.3.1 [Column arithmetic], page 230). For the full equation and basic definitions, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

For example SDSS images are calibrated in units of nanomaggies, with a fixed zero point magnitude of 22.5. Therefore you can convert the units of SDSS image pixels to Janskys with the command below:

```
$ astarithmetic sdss-image.fits 22.5 counts-to-jy
```

**au-to-pc**
Convert Astronomical Units (AUs) to Parsecs (PCs). This operator takes a single argument which is interpreted to be the input AUs. The conversion

---

5 The per-pixel surface brightness limit is the magnitude of the noise standard deviation. For more on surface brightness see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394. In the example command, because the output is a single number, we are using --quiet to avoid printing extra information.
is based on the definition of Parsecs: \(1 \text{PC} = \frac{1}{\tan(1^\prime)} \text{AU}\), where \(1^\prime\) is one arcseconds. In other words, \(1 \text{PC} = \frac{648000}{\pi} \text{AU}\). For example, if we take Pluto’s average distance to the Sun to be 40 AUs, we can obtain its distance in Parsecs using this command:

```
echo 40 | asttable -c'arith $1 au-to-pc'
```

\textbf{pc-to-au} Convert Parsecs (PCs) to Astronomical Units (AUs). This operator takes a single argument which is interpreted to be the input PCs. For more on the conversion equation, see description of \texttt{au-to-pc}. For example, Proxima Centauri (the nearest star to the Solar system) is 1.3020 Parsecs from the Sun, we can calculate this distance in units of AUs with the command below:

```
echo 1.3020 | asttable -c'arith $1 pc-to-au'
```

\textbf{ly-to-pc} Convert Light-years (LY) to Parsecs (PCs). This operator takes a single argument which is interpreted to be the input LYs. The conversion is done from IAU’s definition of the light-year \((9460730472580800 \text{ m} \approx 63241.077 \text{ AU} = 0.306601 \text{ PC}\), for the conversion of AU to PC, see the description of \texttt{au-to-pc}). For example the distance of Andromeda galaxy to our galaxy is 2.5 million light-years, so its distance in kilo-Parsecs can be calculated with the command below (note that we want the output in kilo-parsecs, so we are dividing the output of this operator by 1000):

```
echo 2.5e6 | asttable -c'arith $1 ly-to-pc 1000 /'
```

\textbf{pc-to-ly} Convert Parsecs (PCs) to Light-years (LY). This operator takes a single argument which is interpreted to be the input PCs. For the conversion and an example of the inverse of this operator, see the description of \texttt{ly-to-pc}.

\textbf{ly-to-au} Convert Light-years (LY) to Astronomical Units (AUs). This operator takes a single argument which is interpreted to be the input LYs. For the conversion and a similar example, see the description of \texttt{ly-to-pc}.

\textbf{au-to-ly} Convert Astronomical Units (AUs) to Light-years (LY). This operator takes a single argument which is interpreted to be the input AUs. For the conversion and a similar example, see the description of \texttt{ly-to-pc}.

\subsection{Statistical operators}

The operators in this section take a single dataset as input, and will return the desired statistic as a single value.

\textbf{minvalue} Minimum value in the first popped operand, so “\texttt{a.fits minvalue}” will push the minimum pixel value in this image onto the stack. When this operator acts on a single image, the output (operand that is put back on the stack) will no longer be an image, but a number. The output of this operand is in the same type as the input. This operator is mainly intended for multi-element datasets (for example images or data cubes), if the popped operand is a number, it will just return it without any change.

Note that when the final remaining/output operand is a single number, it is printed onto the standard output. For example with the command below the minimum pixel value in \texttt{image.fits} will be printed in the terminal:

```
$ astarithmetic image.fits minvalue
```
However, the output above also includes a lot of extra information that are not relevant in this context. If you just want the final number, run Arithmetic in quiet mode:

```bash
$ astarithmetic image.fits minvalue -q
```

Also see the description of `sqrt` for other example usages of this operator.

**maxvalue**

Maximum value of first operand in the same type, similar to `minvalue`, see the description there for more. For example

```bash
$ astarithmetic image.fits maxvalue -q
```

**numbervalue**

Number of non-blank elements in first operand in the `uint64` type (since it is always a positive integer, see Section 4.5 [Numeric data types], page 178). Its usage is similar to `minvalue`, for example

```bash
$ astarithmetic image.fits numbervalue -q
```

**sumvalue**

Sum of non-blank elements in first operand in the `float32` type. Its usage is similar to `minvalue`, for example

```bash
$ astarithmetic image.fits sumvalue -q
```

**meanvalue**

Mean value of non-blank elements in first operand in the `float32` type. Its usage is similar to `minvalue`, for example

```bash
$ astarithmetic image.fits meanvalue -q
```

**stdvalue**

Standard deviation of non-blank elements in first operand in the `float32` type. Its usage is similar to `minvalue`, for example

```bash
$ astarithmetic image.fits stdvalue -q
```

**medianvalue**

Median of non-blank elements in first operand with the same type. Its usage is similar to `minvalue`, for example

```bash
$ astarithmetic image.fits medianvalue -q
```

**unique**

Remove all duplicate (and blank) elements from the first popped operand. The unique elements of the dataset will be stored in a single-dimensional dataset. Recall that by default, single-dimensional datasets are stored as a table column in the output. But you can use `--onedasimage` or `--onedonstdout` to respectively store them as a single-dimensional FITS array/image, or to print them on the standard output.

Although you can use this operator on the floating point dataset, due to floating-point errors it may give non-reasonable values: because the tenth digit of the decimal point is also considered although it may be statistically meaningless, see Section 4.5 [Numeric data types], page 178. It is therefore better/recommended to use it on the integer dataset like the labeled images of Section 7.3.1.3 [Segment output], page 390, where each pixel has the integer label of the object/clump it is associated with. For example let’s assume you have cropped a region of a larger labeled image and want to find the labels/objects that are within the crop. With this operator, this job is trivial:

```bash
$ astarithmetic seg-crop.fits unique
```
size  Size of the dataset along a given FITS (or FORTRAN) dimension (counting from 1). The desired dimension should be the first popped operand and the dataset must be the second popped operand. The output will be a single unsigned integer (dimensions cannot be negative). For example, the following command will produce the size of the first extension/HDU (the default HDU) of a.fits along the second FITS axis.

\$ astarithmetic a.fits 2 size

6.2.3.5 Stacking operators

The operators in this section are used when you have multiple datasets that you would like to merge into one, commonly known as “stacking” or “coaddition”. For example, you have taken ten exposures of your scientific target, and you would like to combine them all into one deep stacked image that is deeper.

When calling these operators you should determine how many operands they should take in (unlike the rest of the operators that have a fixed number of input operands). As described in the first operand below, you do this through their first popped operand (which should be a single integer number that is larger than one).

min  For each pixel, find the minimum value in all given datasets. The output will have the same type as the input.

The first popped operand to this operator must be a positive integer number which specifies how many further operands should be popped from the stack. All the subsequently popped operands must have the same type and size. This operator (and all the variable-operand operators similar to it that are discussed below) will work in multi-threaded mode unless Arithmetic is called with the --numthreads=1 option, see Section 4.4 [Multi-threaded operations], page 175.

Each pixel of the output of the min operator will be given the minimum value of the same pixel from all the popped operands/images. For example the following command will produce an image with the same size and type as the three inputs, but each output pixel value will be the minimum of the same pixel’s values in all three input images.

\$ astarithmetic a.fits b.fits c.fits 3 min --output=min.fits

Important notes:
- NaN/blank pixels will be ignored, see Section 6.1.3 [Blank pixels], page 262.
- The output will have the same type as the inputs. This is natural for the min and max operators, but for other similar operators (for example sum, or average) the per-pixel operations will be done in double precision floating point and then stored back in the input type. Therefore, if the input was an integer, C’s internal type conversion will be used.
- The operation will be multi-threaded, greatly speeding up the process if you have large and numerous data to stack. You can disable multi-threaded operations with the --numthreads=1 option (see Section 4.4 [Multi-threaded operations], page 175).

max  For each pixel, find the maximum value in all given datasets. The output will have the same type as the input. This operator is called similar to the min operator, please see there for more. For example
$ astarithmetic a.fits b.fits c.fits 3 max -omax.fits

dnumber For each pixel count the number of non-blank pixels in all given datasets. The output will be an unsigned 32-bit integer datatype (see Section 4.5 [Numeric data types], page 178). This operator is called similar to the min operator, please see there for more. For example

$ astarithmetic a.fits b.fits c.fits 3 number -onum.fits

Some datasets may have blank values (which are also ignored in all similar operators like min, sum, mean or median). Hence, the final pixel values of this operator will not, in general, be equal to the number of inputs. This operator is therefore mostly called in parallel with those operators to know the “weight” of each pixel (in case you want to only keep pixels that had the full exposure for example).

$s astarithmetic a.fits b.fits c.fits 3 number -onum.fits$

sum For each pixel, calculate the sum in all given datasets. The output will have the a single-precision (32-bit) floating point type. This operator is called similar to the min operator, please see there for more. For example

$ astarithmetic a.fits b.fits c.fits 3 sum -ostack-sum.fits

mean For each pixel, calculate the mean in all given datasets. The output will have the a single-precision (32-bit) floating point type. This operator is called similar to the min operator, please see there for more. For example

$ astarithmetic a.fits b.fits c.fits 3 mean -ocoadd-mean.fits

std For each pixel, find the standard deviation in all given datasets. The output will have the a single-precision (32-bit) floating point type. This operator is called similar to the min operator, please see there for more. For example

$ astarithmetic a.fits b.fits c.fits 3 std -ostd.fits

median For each pixel, find the median in all given datasets. The output will have the a single-precision (32-bit) floating point type. This operator is called similar to the min operator, please see there for more. For example

$ astarithmetic a.fits b.fits c.fits 3 median \
   --output=stack-median.fits

quantile For each pixel, find the quantile from all given datasets. The output will have the same numeric data type and size as the input datasets. Besides the input datasets, the quantile operator also needs a single parameter (the requested quantile). The parameter should be the first popped operand, with a value between (and including) 0 and 1. The second popped operand must be the number of datasets to use.

In the example below, the first-popped operand (0.7) is the quantile, the second-popped operand (3) is the number of datasets to pop.

$ astarithmetic a.fits b.fits c.fits 3 0.7 quantile

sigclip-number For each pixel, find the sigma-clipped number (after removing outliers) in all given datasets. The output will have the an unsigned 32-bit integer type (see Section 4.5 [Numeric data types], page 178).
This operator will combine the specified number of inputs into a single output that contains the number of remaining elements after $\sigma$-clipping on each element/pixel (for more on $\sigma$-clipping, see Section 7.1.3 [Sigma clipping], page 343). This operator is very similar to min, with the exception that it expects two operands (parameters for sigma-clipping) before the total number of inputs. The first popped operand is the termination criteria and the second is the multiple of $\sigma$.

For example in the command below, the first popped operand (0.2) is the sigma clipping termination criteria. If the termination criteria is larger than, or equal to, 1 it is interpreted as the number of clips to do. But if it is between 0 and 1, then it is the tolerance level on the standard deviation (see Section 7.1.3 [Sigma clipping], page 343). The second popped operand (5) is the multiple of sigma to use in sigma-clipping. The third popped operand (10) is number of datasets that will be used (similar to the first popped operand to min).

```
astarithmetic a.fits b.fits c.fits 3 5 0.2 sigclip-number
```

**sigclip-median**

For each pixel, find the sigma-clipped median in all given datasets. The output will have the a single-precision (32-bit) floating point type. This operator is called similar to the `sigclip-number` operator, please see there for more. For example

```
astarithmetic a.fits b.fits c.fits 3 5 0.2 sigclip-median
```

**sigclip-mean**

For each pixel, find the sigma-clipped mean in all given datasets. The output will have the a single-precision (32-bit) floating point type. This operator is called similar to the `sigclip-number` operator, please see there for more. For example

```
astarithmetic a.fits b.fits c.fits 3 5 0.2 sigclip-mean
```

**sigclip-std**

For each pixel, find the sigma-clipped standard deviation in all given datasets. The output will have the a single-precision (32-bit) floating point type. This operator is called similar to the `sigclip-number` operator, please see there for more. For example

```
astarithmetic a.fits b.fits c.fits 3 5 0.2 sigclip-std
```

### 6.2.3.6 Filtering (smoothing) operators

Image filtering is commonly used for smoothing: every pixel value in the output image is created by applying a certain statistic to the pixels in its vicinity.

**filter-mean**

Apply mean filtering (or moving average (https://en.wikipedia.org/wiki/Moving_average)) on the input dataset. During mean filtering, each pixel (data element) is replaced by the mean value of all its surrounding pixels (excluding blank values). The number of surrounding pixels in each dimension (to calculate the mean) is determined through the earlier operands that have been pushed onto the stack prior to the input dataset. The number of necessary operands
is determined by the dimensions of the input dataset (first popped operand).
The order of the dimensions on the command-line is the order in FITS format.
Here is one example:

```bash
$ astarithmetic 5 4 image.fits filter-mean
```

In this example, each pixel is replaced by the mean of a 5 by 4 box around it.
The box is 5 pixels along the first FITS dimension (horizontal when viewed in
ds9) and 4 pixels along the second FITS dimension (vertical).
Each pixel will be placed in the center of the box that the mean is calculated on.
If the given width along a dimension is even, then the center is assumed to be
between the pixels (not in the center of a pixel). When the pixel is close to the
edge, the pixels of the box that fall outside the image are ignored. Therefore,
on the edge, less points will be used in calculating the mean.
The final effect of mean filtering is to smooth the input image, it is essentially
a convolution with a kernel that has identical values for all its pixels (is flat),
see Section 6.3.1.1 [Convolution process], page 307.
Note that blank pixels will also be affected by this operator: if there are any
non-blank elements in the box surrounding a blank pixel, in the filtered image,
it will have the mean of the non-blank elements, therefore it won’t be blank
any more. If blank elements are important for your analysis, you can use the
isblank with the where operator to set them back to blank after filtering.

**filter-median**

Apply median filtering (https://en.wikipedia.org/wiki/Median_filter)
on the input dataset. This is very similar to filter-mean, except that instead
of the mean value of the box pixels, the median value is used to replace a pixel
value. For more on how to use this operator, please see filter-mean.
The median is less susceptible to outliers compared to the mean. As a result,
after median filtering, the pixel values will be more discontinuous than mean
filtering.

**filter-sigclip-mean**

Apply a σ-clipped mean filtering onto the input dataset. This is very similar to
filter-mean, except that all outliers (identified by the σ-clipping algorithm)
have been removed, see Section 7.1.3 [Sigma clipping], page 343, for more on
the basics of this algorithm. As described there, two extra input parameters
are necessary for σ-clipping: the multiple of σ and the termination criteria.
filter-sigclip-mean therefore needs to pop two other operands from the
stack after the dimensions of the box.
For example the line below uses the same box size as the example of
filter-mean. However, all elements in the box that are iteratively beyond
3σ of the distribution’s median are removed from the final calculation of the
mean until the change in σ is less than 0.2.

```bash
$ astarithmetic 3 0.2 5 4 image.fits filter-sigclip-mean
```

The median (which needs a sorted dataset) is necessary for σ-clipping, therefore
filter-sigclip-mean can be significantly slower than filter-mean. However,
if there are strong outliers in the dataset that you want to ignore (for example
emission lines on a spectrum when finding the continuum), this is a much better solution.

**filter-sigclip-median**

Apply a $\sigma$-clipped median filtering onto the input dataset. This operator and its necessary operands are almost identical to **filter-sigclip-mean**, except that after $\sigma$-clipping, the median value (which is less affected by outliers than the mean) is added back to the stack.

### 6.2.3.7 Interpolation operators

Interpolation is the process of removing blank pixels from a dataset (by giving them a value based on the non-blank neighbors).

**interpolate-medianngb**

Interpolate the blank elements of the second popped operand with the median of nearest non-blank neighbors to each. The number of the nearest non-blank neighbors used to calculate the median is given by the first popped operand. The distance of the nearest non-blank neighbors is irrelevant in this interpolation. The neighbors of each blank pixel will be parsed in expanding circular rings (for 2D images) or spherical surfaces (for 3D cube) and each non-blank element over them is stored in memory. When the requested number of non-blank neighbors have been found, their median is used to replace that blank element. For example the line below replaces each blank element with the median of the nearest 5 pixels.

```
$ astarithmetic image.fits 5 interpolate-medianngb
```

When you want to interpolate blank regions and you want each blank region to have a fixed value (for example the centers of saturated stars) this operator is not good. Because the pixels used to interpolate various parts of the region differ. For such scenarios, you may use **interpolate-maxofregion** or **interpolate-inofregion** (described below).

**interpolate-minngb**

Similar to **interpolate-medianngb**, but will fill the blank values of the dataset with the minimum value of the nearest neighbors.

**interpolate-maxngb**

Similar to **interpolate-medianngb**, but will fill the blank values of the dataset with the maximum value of the nearest neighbors. One useful implementation of this operator is to fill the saturated pixels of stars in images.

**interpolate-minofregion**

Interpolate all blank regions (consisting of many blank pixels that are touching) in the second popped operand with the minimum value of the pixels that are immediately bordering that region (a single value). The first popped operand is the connectivity (see description in **connected-components**).

For example with the command below all the connected blank regions of **image.fits** will be filled. Its an image (2D dataset), so a 2 connectivity means that the independent blank regions are defined by 8-connected neighbors. If connectivity was 1, the regions would be defined by 4-connectivity: blank
regions that may only be touching on the corner of one pixel would be identified as separate regions.

\$ astarithmetic image.fits 2 interpolate-minofregion

**interpolate-maxofregion**

Similar to `interpolate-minofregion`, but the maximum is used to fill the blank regions.

This operator can be useful in filling saturated pixels in stars for example. Recall that the `interpolate-maxngb` operator looks for the maximum value with a given number of neighboring pixels and is more useful in small noisy regions. Therefore as the blank regions become larger, `interpolate-maxngb` can cause a fragmentation in the connected blank region because the nearest neighbor to one part of the blank region, may not fall within the pixels searched for the other regions. With this option, the size of the blank region is irrelevant: all the pixels bordering the blank region are parsed and their maximum value is used for the whole region.

### 6.2.3.8 Dimensionality changing operators

Through these operators you can change the dimensions of the output through certain statistics on the dimensions that should be removed. For example, let’s assume you have a 3D data cube that has 300 by 300 pixels in the RA and Dec dimensions (first two dimensions), and 3600 slices along the wavelength (third dimension), so the whole cube is $300 \times 300 \times 3600$ voxels (volume elements). To create a narrow-band image that only contains 100 slices around a certain wavelength, you can crop that section (using Section 6.1 [Crop], page 259), giving you a $300 \times 300 \times 100$ cube. You can now use the `collapse-sum` operator below to “collapse” all the 100 slices into one 2D image that has $300 \times 300$ pixels. Every pixel in this 2D image will have the flux of the sum of the 100 slices.

**collapse-sum**

Collapse the given dataset (second popped operand), by summing all elements along the first popped operand (a dimension in FITS standard: counting from one, from fastest dimension). The returned dataset has one dimension less compared to the input.

The output will have a double-precision floating point type irrespective of the input dataset’s type. Doing the operation in double-precision (64-bit) floating point will help the collapse (summation) be affected less by floating point errors. But afterwards, single-precision floating points are usually enough in real (noisy) datasets. So depending on the type of the input and its nature, it is recommended to use one of the type conversion operators on the returned dataset.

If any WCS is present, the returned dataset will also lack the respective dimension in its WCS matrix. Therefore, when the WCS is important for later processing, be sure that the input is aligned with the respective axes: all non-diagonal elements in the WCS matrix are zero.

One common application of this operator is the creation of pseudo broad-band or narrow-band 2D images from 3D data cubes. For example integral field unit (IFU) data products that have two spatial dimensions (first two FITS
dimensions) and one spectral dimension (third FITS dimension). The command below will collapse the whole third dimension into a 2D array the size of the first two dimensions, and then convert the output to single-precision floating point (as discussed above).

$$\texttt{astarithmetic cube.fits 3 collapse-sum float32}$$

collapse-mean
Similar to collapse-sum, but the returned dataset will be the mean value along the collapsed dimension, not the sum.

collapse-number
Similar to collapse-sum, but the returned dataset will be the number of non-blank values along the collapsed dimension. The output will have a 32-bit signed integer type. If the input dataset doesn’t have blank values, all the elements in the returned dataset will have a single value (the length of the collapsed dimension). Therefore this is mostly relevant when there are blank values in the dataset.

collapse-min
Similar to collapse-sum, but the returned dataset will have the same numeric type as the input and will contain the minimum value for each pixel along the collapsed dimension.

collapse-max
Similar to collapse-sum, but the returned dataset will have the same numeric type as the input and will contain the maximum value for each pixel along the collapsed dimension.

add-dimension-slow
Build a higher-dimensional dataset from all the input datasets stacked after one another (along the slowest dimension). The first popped operand has to be a single number. It is used by the operator to know how many operands it should pop from the stack (and the size of the output in the new dimension). The rest of the operands must have the same size and numerical data type. This operator currently only works for 2D input operands, please contact us if you want inputs to have different dimensions.

The output’s WCS (which should have a different dimensionality compared to the inputs) can be read from another file with the --wcsfile option. If no file is specified for the WCS, the first dataset’s WCS will be used, you can later add/change the necessary WCS keywords with the FITS keyword modification features of the Fits program (see Section 5.1 [Fits], page 194).

If your datasets don’t have the same type, you can use the type transformation operators of Arithmetic that are discussed below. Just beware of overflow if you are transforming to a smaller type, see Section 4.5 [Numeric data types], page 178.

For example, let’s assume you have 3 two-dimensional images a.fits, b.fits and c.fits (each with 200 × 100 pixels). You can construct a 3D data cube with 200 × 100 × 3 voxels (volume-pixels) using the command below:

$$\texttt{astarithmetic a.fits b.fits c.fits 3 add-dimension-slow}$$
add-dimension-fast
Similar to add-dimension-slow but along the fastest dimension. This operator currently only works for 1D input operands, please contact us if you want inputs to have different dimensions.

For example, let’s assume you have 3 one-dimensional datasets, each with 100 elements. With this operator, you can construct a $3 \times 100$ pixel FITS image that has 3 pixels along the horizontal and 5 pixels along the vertical.

6.2.3.9 Conditional operators

Conditional operators take two inputs and return a binary output that can only have two values 0 (for pixels where the condition was false) or 1 (for the pixels where the condition was true). Because of the binary (2-valued) nature of their outputs, the output is therefore stored in an unsigned char data type (see Section 4.5 [Numeric data types], page 178) to speed up process and take less space in your storage. There are two exceptions to the general features above: isblank only takes one input, and where takes three, while not returning a binary output, see their description for more.

lt
Less than: creates a binary output (values either 0 or 1) where each pixel will be 1 if the second popped operand is smaller than the first popped operand and 0 otherwise. If both operands are images, then all the pixels will be compared with their counterparts in the other image.

For example, the pixels in the output of the command below will have a value of 1 (true) if their value in image1.fits is less than their value in image2.fits. Otherwise, their value will be 0 (false).

```bash
$ astarithmetic image1.fits image2.fits lt
```

If only one operand is an image, then all the pixels will be compared with the single value (number) of the other operand. For example:

```bash
$ astarithmetic image1.fits 1000 lt
```

Finally if both are numbers, then the output is also just one number (0 or 1).

```bash
$ astarithmetic 4 5 lt
```

le
Less or equal: similar to lt (‘less than’ operator), but returning 1 when the second popped operand is smaller or equal to the first. For example

```bash
$ astarithmetic image1.fits 1000 le
```

gt
Greater than: similar to lt (‘less than’ operator), but returning 1 when the second popped operand is greater than the first. For example

```bash
$ astarithmetic image1.fits 1000 gt
```

ge
Greater or equal: similar to lt (‘less than’ operator), but returning 1 when the second popped operand is larger or equal to the first. For example

```bash
$ astarithmetic image1.fits 1000 ge
```

eye
Equality: similar to lt (‘less than’ operator), but returning 1 when the two popped operands are equal (to double precision floating point accuracy).

```bash
$ astarithmetic image1.fits 1000 eq
```
Non-Equality: similar to `lt` (‘less than’ operator), but returning 1 when the two popped operands are not equal (to double precision floating point accuracy).

```bash
$ astarithmetic image1.fits 1000 ne
```

Logical AND: returns 1 if both operands have a non-zero value and 0 if both are zero. Both operands have to be the same kind: either both images or both numbers and it mostly makes meaningful values when the inputs are binary (with pixel values of 0 or 1).

```bash
$ astarithmetic image1.fits image2.fits -g1 and
```

For example if you only want to see which pixels in an image have a value between 50 (greater equal, or inclusive) and 200 (less than, or exclusive), you can use this command:

```bash
$ astarithmetic image.fits set-i i 50 ge i 200 lt and
```

Logical OR: returns 1 if either one of the operands is non-zero and 0 only when both operators are zero. Both operands have to be the same kind: either both images or both numbers. The usage is similar to `and`.

For example if you only want to see which pixels in an image have a value outside of -100 (greater equal, or inclusive) and 200 (less than, or exclusive), you can use this command:

```bash
$ astarithmetic image.fits set-i i -100 lt i 200 ge or
```

Logical NOT: returns 1 when the operand is 0 and 0 when the operand is non-zero. The operand can be an image or number, for an image, it is applied to each pixel separately. For example if you want to know which pixels are not blank, you can use `not` on the output of the `isblank` operator described below:

```bash
$ astarithmetic image.fits isblank not
```

Test for a blank value (see Section 6.1.3 [Blank pixels], page 262). In essence, this is very similar to the conditional operators: the output is either 1 or 0 (see the ‘less than’ operator above). The difference is that it only needs one operand. For example:

```bash
$ astarithmetic image.fits isblank
```

Because of the definition of a blank pixel, a blank value is not even equal to itself, so you cannot use the equal operator above to select blank pixels. See the “Blank pixels” box below for more on Blank pixels in Arithmetic.

Change the input (pixel) value `where`/if a certain condition holds. The conditional operators above can be used to define the condition. Three operands are required for `where`. The input format is demonstrated in this simplified example:

```bash
$ astarithmetic modify.fits binary.fits if-true.fits where
```

The value of any pixel in `modify.fits` that corresponds to a non-zero and non-blank pixel of `binary.fits` will be changed to the value of the same pixel in `if-true.fits` (this may also be a number). The 3rd and 2nd popped operands (`modify.fits` and `binary.fits` respectively, see Section 6.2.1 [Reverse polish notation], page 272) have to have the same dimensions/size. `if-true.fits` can be either a number, or have the same dimension/size as the other two.
The 2nd popped operand (binary.fits) has to have uint8 (or unsigned char in standard C) type (see Section 4.5 [Numeric data types], page 178). It is treated as a binary dataset (with only two values: zero and non-zero, hence the name binary.fits in this example). However, commonly you won’t be dealing with an actual FITS file of a condition/binary image. You will probably define the condition in the same run based on some other reference image and use the conditional and logical operators above to make a true/false (or one/zero) image for you internally. For example the case below:

\$ astarithmetic in.fits reference.fits 100 gt new.fits where

In the example above, any of the in.fits pixels that has a value in reference.fits greater than 100, will be replaced with the corresponding pixel in new.fits. Effectively the reference.fits 100 gt part created the condition/binary image which was added to the stack (in memory) and later used by where. The command above is thus equivalent to these two commands:

\$ astarithmetic reference.fits 100 gt --output=binary.fits
\$ astarithmetic in.fits binary.fits new.fits where

Finally, the input operands are read and used independently, so you can use the same file more than once as any of the operands.

When the 1st popped operand to where (if-true.fits) is a single number, it may be a NaN value (or any blank value, depending on its type) like the example below (see Section 6.1.3 [Blank pixels], page 262). When the number is blank, it will be converted to the blank value of the type of the 3rd popped operand (in.fits). Hence, in the example below, all the pixels in reference.fits that have a value greater than 100, will become blank in the natural data type of in.fits (even though NaN values are only defined for floating point types).

\$ astarithmetic in.fits reference.fits 100 gt nan where

### 6.2.3.10 Mathematical morphology operators

From Wikipedia: “Mathematical morphology (MM) is a theory and technique for the analysis and processing of geometrical structures, based on set theory, lattice theory, topology, and random functions. MM is most commonly applied to digital images”. In theory it extends a very large body of research and methods in image processing, but currently in Gnuastro it mainly applies to images that are binary (only have a value of 0 or 1). For example you have applied the greater-than operator (gt, see Section 6.2.3.9 [Conditional operators], page 290) to select all pixels in your image that are larger than a value of 100. But they will all have a value of 1, and you want to separate the various groups of pixels that are connected (for example peaks of stars in your image). With the connected-components operator, you can give each connected region of the output of gt a separate integer label.

**erode**

Erode the foreground pixels (with value 1) of the input dataset (second popped operand). The first popped operand is the connectivity (see description in connected-components). Erosion is simply a flipping of all foreground pixels (with value 1) to background (with value 0) that are “touching” background pixels. “Touching” is defined by the connectivity.
In effect, this operator “carves off” the outer borders of the foreground, making them thinner. This operator assumes a binary dataset (all pixels are 0 or 1). For example, imagine that you have an astronomical image with a mean/sky value of 0 units and a standard deviation ($\sigma$) of 100 units and many galaxies in it. With the first command below, you can apply a threshold of $2\sigma$ on the image (by only keeping pixels that are greater than 200 using the $gt$ operator). The output of thresholding the image is a binary image (each pixel is either smaller or equal to the threshold or larger than it). You can then erode the binary image with the second command below to remove very small false positives (one or two pixel peaks).

```
$ astarithmetic image.fits 100 gt -obinary.fits
$ astarithmetic binary.fits 2 erode -oout.fits
```

In fact, you can merge these operations into one command thanks to the reverse polish notation (see Section 6.2.1 [Reverse polish notation], page 272):

```
$ astarithmetic image.fits 100 gt 2 erode -oout.fits
```

To see the effect of connectivity, try this:

```
$ astarithmetic image.fits 100 gt 1 erode -oout-con-1.fits
```

dilate

Dilate the foreground pixels (with value 1) of the binary input dataset (second popped operand). The first popped operand is the connectivity (see description in connected-components). Dilation is simply a flipping of all background pixels (with value 0) to foreground (with value 1) that are “touching” foreground pixels. “Touching” is defined by the connectivity. In effect, this expands the outer borders of the foreground. This operator assumes a binary dataset (all pixels are 0 and 1). The usage is similar to erode, for example:

```
$ astarithmetic binary.fits 2 dilate -oout.fits
```

connected-components

Find the connected components in the input dataset (second popped operand). The first popped is the connectivity used in the connected components algorithm. The second popped operand is the dataset where connected components are to be found. It is assumed to be a binary image (with values of 0 or 1). It must have an 8-bit unsigned integer type which is the format produced by conditional operators. This operator will return a labeled dataset where the non-zero pixels in the input will be labeled with a counter (starting from 1). The connectivity is a number between 1 and the number of dimensions in the dataset (inclusive). 1 corresponds to the weakest (symmetric) connectivity between elements and the number of dimensions the strongest. For example on a 2D image, a connectivity of 1 corresponds to 4-connected neighbors and 2 corresponds to 8-connected neighbors.

One example usage of this operator can be the identification of regions above a certain threshold, as in the command below. With this command, Arithmetic will first separate all pixels greater than 100 into a binary image (where pixels with a value of 1 are above that value). Afterwards, it will label all those that are connected.

```
$ astarithmetic in.fits 100 gt 2 connected-components
```
If your input dataset doesn't have a binary type, but you know all its values are 0 or 1, you can use the uint8 operator (below) to convert it to binary.

**fill-holes**

Flip background (0) pixels surrounded by foreground (1) in a binary dataset. This operator takes two operands (similar to connected-components): the second is the binary (0 or 1 valued) dataset to fill holes in and the first popped operand is the connectivity (to define a hole). Imagine that in your dataset there are some holes with zero value inside the objects with one value (for example the output of the thresholding example of erode) and you want to fill the holes:

```
$ astarithmetic binary.fits 2 fill-holes
```

**invert**

Invert an unsigned integer dataset (won’t work on other data types, see Section 4.5 [Numeric data types], page 178). This is the only operator that ignores blank values (which are set to be the maximum values in the unsigned integer types).

This is useful in cases where the target(s) has(have) been imaged in absorption as raw formats (which are unsigned integer types). With this option, the maximum value for the given type will be subtracted from each pixel value, thus “inverting” the image, so the target(s) can be treated as emission. This can be useful when the higher-level analysis methods/tools only work on emission (positive skew in the noise, not negative).

```
$ astarithmetic image.fits invert
```

## 6.2.3.11 Bitwise operators

Astronomical images are usually stored as an array multi-byte pixels with different sizes for different precision levels (see Section 4.5 [Numeric data types], page 178). For example images from CCDs are usually in the unsigned 16-bit integer type (each pixel takes 16 bits, or 2 bytes, of memory) and fully reduced deep images have a 32-bit floating point type (each pixel takes 32 bits or 4 bytes).

On the other hand, during the data reduction, we need to preserve a lot of meta-data about some pixels. For example, if a cosmic ray had hit the pixel during the exposure, or if the pixel was saturated, or is known to have a problem, or if the optical vignetting is too strong on it, and etc. A crude solution is to make a new image when checking for each one of these things and make a binary image where we flag (set to 1) pixels that satisfy any of these conditions above, and set the rest to zero. However, processing pipelines sometimes need more than 20 flags to store important per-pixel meta-data, and recall that the smallest numeric data type is one byte (or 8 bits, that can store up to 256 different values), while we only need two values for each flag! This is a major waste of storage space!

A much more optimal solution is to use the bits within each pixel to store different flags! In other words, if you have an 8-bit pixel, use each bit as a flag to mark if a certain condition has happened on a certain pixel or not. For example, let’s set the following standard based on the four cases mentioned above: the first bit will show that a cosmic ray has hit that pixel. So if a pixel is only affected by cosmic rays, it will have this sequence of bits (note that the bit-counting starts from the right): 00000001. The second bit shows that the pixel
was saturated (00000010), the third bit shows that it has known problems (00000100) and the fourth bit shows that it was affected by vignetting (00001000).

Since each bit is independent, we can thus mark multiple metadata about that pixel in the actual image, within a single “flag” or “mask” pixel of a flag or mask image that has the same number of pixels. For example a flag-pixel with the following bits 00001001 shows that it has been affected by cosmic rays and it has been affected by vignetting at the same time. The common data type to store these flagging pixels are unsigned integer types (see Section 4.5 [Numeric data types], page 178). Therefore when you open an unsigned 8-bit flag image in a viewer like DS9, you will see a single integer in each pixel that actually has 8 layers of metadata in it! For example the integer you will see for the bit sequences given above will respectively be: $2^0 = 1$ (for a pixel that only has cosmic ray), $2^1 = 2$ (for a pixel that was only saturated), $2^2 = 4$ (for a pixel that only has known problems), $2^3 = 8$ (for a pixel that is only affected by vignetting) and $2^0 + 2^3 = 9$ (for a pixel that has a cosmic ray and was affected by vignetting).

You can later use this bit information to mark objects in your final analysis or to mask certain pixels. For example you may want to set all pixels affected by vignetting to NaN, but can interpolate over cosmic rays. You therefore need ways to separate the pixels with a desired flag(s) from the rest. It is possible to treat a flag pixel as a single integer (and try to define certain ranges in value to select certain flags). But a much more easier and robust way is to actually look at each pixel as a sequence of bits (not as a single integer!) and use the bitwise operators below for this job. For more on the theory behind bitwise operators, see Wikipedia (https://en.wikipedia.org/wiki/Bitwise_operation).

- **bitand**: Bitwise AND operator: only bits with values of 1 in both popped operands will get the value of 1, the rest will be set to 0. For example (assuming numbers can be written as bit strings on the command-line): 00101000 00100010 bitand will give 00100000. Note that the bitwise operators only work on integer type datasets.

- **bitor**: Bitwise inclusive OR operator: The bits where at least one of the two popped operands has a 1 value get a value of 1, the others 0. For example (assuming numbers can be written as bit strings on the command-line): 00101000 00100010 bitand will give 00101010. Note that the bitwise operators only work on integer type datasets.

- **bitxor**: Bitwise exclusive OR operator: A bit will be 1 if it differs between the two popped operands. For example (assuming numbers can be written as bit strings on the command-line): 00101000 00100010 bitand will give 00000010. Note that the bitwise operators only work on integer type datasets.

- **lshift**: Bitwise left shift operator: shift all the bits of the first operand to the left by a number of times given by the second operand. For example (assuming numbers can be written as bit strings on the command-line): 00101000 00100010 bitand will give 00001010. Note that the bitwise operators only work on integer type datasets.

- **rshift**: Bitwise right shift operator: shift all the bits of the first operand to the right by a number of times given by the second operand. For example (assuming numbers can be written as bit strings on the command-line): 00101000 00100010 bitand will give 00001000. This is equivalent to multiplication by 4. Note that the bitwise operators only work on integer type datasets.
give 0001010. Note that the bitwise operators only work on integer type datasets.

**bitnot**  
Bitwise not (more formally known as one’s complement) operator: flip all the bits of the popped operand (note that this is the only unary, or single operand, bitwise operator). In other words, any bit with a value of 0 is changed to 1 and vice-versa. For example (assuming numbers can be written as bit strings on the command-line): 00101000 bitnot will give 11010111. Note that the bitwise operators only work on integer type datasets/numbers.

### 6.2.3.12 Numerical type conversion operators

With the operators below you can convert the numerical data type of your input, see Section 4.5 [Numeric data types], page 178. Type conversion is particularly useful when dealing with integers, see Section 6.2.2 [Integer benefits and pitfalls], page 274.

As an example, let’s assume that your colleague gives you many single exposure images for processing, but they have a double-precision floating point type! You know that the statistical error a single-exposure image can never exceed 6 or 7 significant digits, so you would prefer to archive them as a single-precision floating point and save space on your computer (a double-precision floating point is also double the file size!). You can do this with the **float32** operator described below.

**uint8**  
Convert the type of the popped operand to 8-bit unsigned integer type (see Section 4.5 [Numeric data types], page 178). The internal conversion of C will be used.

**int8**  
Convert the type of the popped operand to 8-bit signed integer type (see Section 4.5 [Numeric data types], page 178). The internal conversion of C will be used.

**uint16**  
Convert the type of the popped operand to 16-bit unsigned integer type (see Section 4.5 [Numeric data types], page 178). The internal conversion of C will be used.

**int16**  
Convert the type of the popped operand to 16-bit signed integer (see Section 4.5 [Numeric data types], page 178). The internal conversion of C will be used.

**uint32**  
Convert the type of the popped operand to 32-bit unsigned integer type (see Section 4.5 [Numeric data types], page 178). The internal conversion of C will be used.

**int32**  
Convert the type of the popped operand to 32-bit signed integer type (see Section 4.5 [Numeric data types], page 178). The internal conversion of C will be used.

**uint64**  
Convert the type of the popped operand to 64-bit unsigned integer (see Section 4.5 [Numeric data types], page 178). The internal conversion of C will be used.

**float32**  
Convert the type of the popped operand to 32-bit (single precision) floating point (see Section 4.5 [Numeric data types], page 178). The internal conversion of C will be used. For example if **f64.fits** is a 64-bit floating point image, and you want to store it as a 32-bit floating point image, you can use the command
below (the second command is to show that the output file consumes half the storage)

```
$ astarithmetic f64.fits float32 --output=f32.fits
$ ls -lh f64.fits f32.fits
```

**float64** Convert the type of the popped operand to 64-bit (double precision) floating point (see Section 4.5 [Numeric data types], page 178). The internal conversion of C will be used.

### 6.2.3.13 Adding noise operators

When you simulate data (for example see Section 2.1 [Sufi simulates a detection], page 18), everything is ideal and there is no noise! The final step of the process is to add simulated noise to the data. The operators in this section are designed for that purpose.

**mknoise-sigma**

Add a fixed noise (Gaussian standard deviation) to each element of the input dataset. This operator takes two arguments: the top/first popped operand is the noise standard deviation, the next popped operand is the dataset that the noise should be added to.

When `--quiet` isn't given, a statement will be printed on each invocation of this operator (if there are multiple calls to the `mknoise-*`, the statement will be printed multiple times). It will show the random number generator function and seed that was used in that invocation, see Section 8.2.1.4 [Generating random numbers], page 464. Reproducibility of the outputs can be ensured with the `--envseed` option, see below for more.

For example with the first command below, `image.fits` will be degraded by a noise of standard deviation 3 units.

```
$ astarithmetic image.fits 3 mknoise-sigma
```

Alternatively, you can use this operator within column arithmetic in the Table program, to generate a random number like below (centered on 0, with $\sigma = 3$) like the first command below. With the second command, you can put it into a shell variable for later usage.

```
$ echo 0 | asttable -c 'arith $1 3 mknoise-sigma'
$ value=$(echo 0 | asttable -c 'arith $1 3 mknoise-sigma')
$ echo $value
```

You can also use this operator in combination with AWK to easily generate an arbitrarily large table with random columns. In the example below, we'll create a two column table with 20 rows. The first column will be centered on 5 and $\sigma_1 = 2$, the second will be centered on 10 and $\sigma_2 = 3$:

```
$ echo 5 10 \n  | awk '{for(i=0;i<20;++i) print $1, $2}' \n  | asttable -c 'arith $1 2 mknoise-sigma' \n  -c 'arith $2 3 mknoise-sigma'
```

By adding an extra `--output=random.fits`, the table will be saved into a file called `random.fits`, and you can change the i<20 to i<5000 to have 5000 rows instead. Of course, if your input table has different values in the desired column
the noisy distribution will be centered on each input element, but all will have the same scatter/sigma.

You can use the `--envseed` option to fix the random number generator seed (and thus get a reproducible result). For more on `--envseed`, see Section 8.2.1.4 [Generating random numbers], page 464. When using column arithmetic in Table, it may happen that multiple columns need random numbers (with any of the `mknoise-*` operators) in one call of `asttable`. In such cases, the value given to `GSL_RNG_SEED` is incremented by one on every call to the `mknoise-*` operators. Without this increment, when the column values are the same (happens a lot, for no-noised datasets), the returned values for all columns will be identical. But this feature has a side-effect: that if the order of calling the `mknoise-*` operators changes, the seeds used for each operator will change\(^\text{6}\).

**`mknoise-poisson`**

Add Poisson noise to each element of the input dataset (see Section 8.2.1.1 [Photon counting noise], page 462). This operator takes two arguments: the top/first popped operand is the background value (in units of electron counts), the next popped operand is the dataset that the noise should be added to.

Except for the noise-model, this operator is very similar to `mknoise-sigma` and the examples there apply here too. The main difference with `mknoise-sigma` is that in a Poisson distribution the scatter/sigma will depend on each element’s value.

For example, let’s assume you have made a mock image called `mock.fits` with Section 8.1 [MakeProfiles], page 440, and its assumed zero point is 22.5 (for more on the zero point, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394). Let’s assume the background level for the Poisson noise has a value of 19 magnitudes. You can first use the `mag-to-counts` operator to convert this background magnitude into counts, then feed the background value in counts to `mknoise-poisson` operator:

```bash
$ astarithmetic mock.fits 19 22.5 mag-to-counts \mknoise-poisson
```

Try changing the background value from 19 to 10 to see the effect! Recall that the tutorial Section 2.1 [Sufi simulates a detection], page 18, shows how you can use MakeProfiles to build mock images.

**`mknoise-uniform`**

Add uniform noise to each element of the input dataset. This operator takes two arguments: the top/first popped operand is the width of the interval, the second popped operand is the dataset that the noise should be added to (each element will be the center of the interval). The returned random values may happen to be the minimum interval value, but will never be the maximum. Except for the noise-model, this operator behaves very similar to `mknoise-sigma`, see the explanation there for more.

\(^6\) We have defined Task 15971 ([https://savannah.gnu.org/task/?15971](https://savannah.gnu.org/task/?15971)) in Gnuastro’s project management system to address this. If you need this feature please send us an email at bug-gnuastro@gnu.org (to motivate us in its implementation).
For example with the command below, a random value will be selected between 10 to 14 (centered on 12, which is the only input data element, with a total width of 4).

```
  echo 12 | asttable -c 'arith $1 4 mknoise-uniform'
```

Similar to the example in `mknoise-sigma`, you can pipe the output of `echo` to `awk` before passing it to `asttable` to generate a full column of uniformly selected values within the same interval.

### 6.2.3.14 Elliptical shape operators

The operators here describe certain functions that will be necessary when dealing with objects that have a certain elliptical shape.

**box-around-ellipse**

Return the width (along horizontal) and height (along vertical) of a box that encompasses an ellipse with the same center point. The top-popped operand is assumed to be the position angle (angle from the horizontal axis) in degrees. The second and third popped operands are the minor and major axis lengths respectively. This operator outputs two operands on the general stack. The first one is the width and the second (which will be the top one when this operator finishes) is the height.

If the value to the second popped operand (minor axis) is larger than the third (major axis), a NaN value will be written for both the width and height of that element and a warning will be printed (the warning can be disabled with the `--quiet` option).

As an example, if your ellipse has a major axis length of 10 units, a minor axis length of 4 units and a position angle of 20 degrees, you can estimate the bounding box with this command:

```
  $ echo "10 4 20" \
       | asttable -c 'arith $1 $2 $3 box-around-ellipse'
```

Alternatively if your three values are in separate FITS arrays/images, you can use the command below to have the width and height in similarly sized fits arrays. In this example `a.fits` and `b.fits` are respectively the major and minor axis lengths and `pa.fits` is the position angle (in degrees). Also, in all three, we assume the first extension is used. After its done, the height of the box will be put in `h.fits` and the width will be in `w.fits`. Just note that because this operator has two output datasets, you need to first write the height (top output operand) into a file and free it with the `tofilefree` operator, then write the width in the file given to `--output`.

```
  $ astarithmetic a.fits b.fits pa.fits box-around-ellipse \       
      tofilefree-h.fits -ow.fits -g1
```

Finally, if you need to treat the width and height separately for further processing, you can call the `set-` operator two times afterwards like below. Recall that the `set-` operator will pop the top operand, and put it in memory with a certain name, bringing the next operand to the top of the stack.

For example let’s assume `catalog.fits` has at least three columns `MAJOR`, `MINOR` and `PA` which specify the major axis, minor axis and position angle
respectively. But you want the final width and height in 32-bit floating point numbers (not the default 64-bit, which may be too much precision in many scenarios). You can do this with the command below (note you can also break lines with \, within the single-quote environment)

```
$ asttable catalog.fits \
   -c 'arith MAJOR MINOR PA box-around-ellipse \n      set-height set-width \n      width float32 height float32'
```

6.2.3.15 Building new dataset

With the operator here, you can create a new dataset from scratch to start certain operations without any input data.

**makenew** Create a new dataset that only has zero values. The number of dimensions is read as the first popped operand and the number of elements along each dimension are the next popped operand (in reverse of the popping order). The type of the new dataset is an unsigned 8-bit integer and all pixel values have a value of zero. For example, if you want to create a new 100 by 200 pixel image, you can run this command:

```
$ astarithmetic 100 200 2 makenew
```

To further extend the example, you can use any of the noise-making operators to add noise to this new dataset (see Section 6.2.3.13 [Adding noise operators], page 297), like the command below:

```
$ astarithmetic 100 200 2 makenew 5 mknoise-sigma
```

6.2.3.16 Operand storage in memory or a file

In your early days of using Gnuastro, to do multiple operations, it is likely that you will simply call Arithmetic (or Table, with column arithmetic) multiple times: feed the output file of the first call to the second call. But as you get more proficient in the reverse polish notation, you will find yourself combining many operations into one call. This greatly speeds up your operation, because instead of writing the dataset to a file in one command, and reading it in the next command, it will just keep the intermediate dataset in memory!

But adding more complexity to your operations, can make them much harder to debug, or extend even further. Therefore in this section we have some special operators that behave differently from the rest: they don’t touch the contents of the data, only where/how they are stored. They are designed to do complex operations, without necessarily having a complex command.

**set-AAA** Set the characters after the dash (AAA in the case shown here) as a name for the first popped operand on the stack. The named dataset will be freed from memory as soon as it is no longer needed, or if the name is reset to refer to another dataset later in the command. This operator thus enables re-usability of a dataset without having to re-read it from a file every time it is necessary during a process. When a dataset is necessary more than once, this operator can thus help simplify reading/writing on the command-line (thus avoiding potential bugs), while also speeding up the processing.
Like all operators, this operator pops the top operand off of the main processing stack, but unlike other operands, it won't add anything back to the stack immediately. It will keep the popped dataset in memory through a separate list of named datasets (not on the main stack). That list will be used to add/copy any requested dataset to the main processing stack when the name is called.

The name to give the popped dataset is part of the operator's name. For example the set-a operator of the command below, gives the name “a” to the contents of image.fits. This name is then used instead of the actual filename to multiply the dataset by two.

```
$ astarithmetic image.fits set-a a 2 x
```

The name can be any string, but avoid strings ending with standard filename suffixes (for example .fits).

One example of the usefulness of this operator is in the where operator. For example, let's assume you want to mask all pixels larger than 5 in image.fits (extension number 1) with a NaN value. Without setting a name for the dataset, you have to read the file two times from memory in a command like this:

```
$ astarithmetic image.fits image.fits 5 gt nan where -g1
```

But with this operator you can simply give image.fits the name i and simplify the command above to the more readable one below (which greatly helps when the filename is long):

```
$ astarithmetic image.fits set-i i i 5 gt nan where
```

**repeat**

Add N copies of the second popped operand to the stack of operands. N is the first popped operand. For example, let's assume image.fits is a 100 × 100 image. The output of the command below will be a 3D datacube of size 100 × 100 × 20 voxels (volume-pixels):

```
$ astarithmetic image.fits 20 repeat 20 add-dimension-slow
```

**tofile-AAA**

Write the top operand on the operands stack into a file called AAA (can be any FITS file name) without changing the operands stack. If you don't need the dataset any more and would like to free it, see the tofilefree operator below.

By default, any file that is given to this operator is deleted before Arithmetic actually starts working on the input datasets. The deletion can be deactivated with the --dontdelete option (as in all Gnuastro programs, see Section 4.1.2.1 [Input/Output options], page 157). If the same FITS file is given to this operator multiple times, it will contain multiple extensions (in the same order that it was called.

For example the operator tofile-check.fits will write the top operand to check.fits. Since it doesn't modify the operands stack, this operator is very convenient when you want to debug, or understanding, a string of operators and operands given to Arithmetic: simply put tofile-AAA anywhere in the

---

7 A dataset name like a.fits (which can be set with set-a.fits) will cause confusion in the initial parser of Arithmetic. It will assume this name is a FITS file, and if it is used multiple times, Arithmetic will abort, complaining that you haven’t provided enough HDUs.
process to see what is happening behind the scenes without modifying the overall process.

tofilefree-AAA

Similar to the tofile operator, with the only difference that the dataset that is written to a file is popped from the operand stack and freed from memory (cannot be used any more).

6.2.4 Invoking Arithmetic

Arithmetic will do pixel to pixel arithmetic operations on the individual pixels of input data and/or numbers. For the full list of operators with explanations, please see Section 6.2.3 [Arithmetic operators], page 276. Any operand that only has a single element (number, or single pixel FITS image) will be read as a number, the rest of the inputs must have the same dimensions. The general template is:

$ astarithmetic [OPTION...] ASTRdata1 [ASTRdata2] OPERATOR ...

One line examples:

## Calculate (10.32-3.84)^2.7 quietly (will just print 155.329):
$ astarithmetic -q 10.32 3.84 - 2.7 pow

## Inverse the input image (1/pixel):
$ astarithmetic 1 image.fits / --out=inverse.fits

## Multiply each pixel in image by -1:
$ astarithmetic image.fits -1 x --out=negative.fits

## Subtract extension 4 from extension 1 (counting from zero):
$ astarithmetic image.fits image.fits - --out=skysub.fits --hdu=1 --hdu=4

## Add two images, then divide them by 2 (2 is read as floating point):  
## Note that without the '.0', the '2' will be read/used as an integer.
$ astarithmetic image1.fits image2.fits + 2.0 / --out=average.fits

## Use Arithmetic's average operator:
$ astarithmetic image1.fits image2.fits average --out=average.fits

## Calculate the median of three images in three separate extensions:
$ astarithmetic img1.fits img2.fits img3.fits median -h0 -h1 -h2 --out=median.fits

Arithmetic's notation for giving operands to operators is fully described in Section 6.2.1 [Reverse polish notation], page 272. The output dataset is last remaining operand on the stack. When the output dataset a single number, it will be printed on the command-line. When the output is an array, it will be stored as a file.

The name of the final file can be specified with the --output option, but if its not given, Arithmetic will use “automatic output” on the name of the first FITS image encountered to generate an output file name, see Section 4.9 [Automatic output], page 191. By default, if the
output file already exists, it will be deleted before Arithmetic starts operation. However, this can be disabled with the `--dontdelete` option (see below). At any point during Arithmetic’s operation, you can also write the top operand on the stack to a file, using the `tofile` or `tofilefree` operators, see Section 6.2.3 [Arithmetic operators], page 276.

By default, the world coordinate system (WCS) information of the output dataset will be taken from the first input image (that contains a WCS) on the command-line. This can be modified with the `--wcsfile` and `--wcshdu` options described below. When the `--quiet` option isn’t given, the name and extension of the dataset used for the output’s WCS is printed on the command-line.

Through operators like those starting with `collapse-`, the dimensionality of the inputs may not be the same as the outputs. By default, when the output is 1D, Arithmetic will write it as a table, not an image/array. The format of the output table (plain text or FITS ASCII or binary) can be set with the `--tableformat` option, see Section 4.1.2.1 [Input/Output options], page 157). You can disable this feature (write 1D arrays as FITS images/arrays, or to the standard output) with the `--onedasimage` or `--onedonstdout` options.

See Section 4.1.2 [Common options], page 156, for a review of the options in all Gnuastro programs. Arithmetic just redefines the `--hdu` and `--dontdelete` options as explained below.

-h INT/STR
`--hdu INT/STR`

The header data unit of the input FITS images, see Section 4.1.2.1 [Input/Output options], page 157. Unlike most options in Gnuastro (which will ultimately only have one value for this option), Arithmetic allows `--hdu` to be called multiple times and the value of each invocation will be stored separately (for the unlimited number of input images you would like to use). Recall that for other programs this (common) option only takes a single value. So in other programs, if you specify it multiple times on the command-line, only the last value will be used and in the configuration files, it will be ignored if it already has a value.

The order of the values to `--hdu` has to be in the same order as input FITS images. Options are first read from the command-line (from left to right), then top-down in each configuration file, see Section 4.2.2 [Configuration file precedence], page 170.

If the number of HDUs is less than the number of input images, Arithmetic will abort and notify you. However, if there are more HDUs than FITS images, there is no problem: they will be used in the given order (every time a FITS image comes up on the stack) and the extra HDUs will be ignored in the end. So there is no problem with having extra HDUs in the configuration files and by default several HDUs with a value of 0 are kept in the system-wide configuration file when you install Gnuastro.

-g INT/STR
`--globalhdu INT/STR`

Use the value to this option as the HDU of all input FITS files. This option is very convenient when you have many input files and the dataset of interest is
in the same HDU of all the files. When this option is called, any values given to the --hdu option (explained above) are ignored and will not be used.

-w FITS
--wcsfile FITS
FITS Filename containing the WCS structure that must be written to the output. The HDU/extension should be specified with --wcshdu.

When this option is used, the respective WCS will be read before any processing is done on the command-line and directly used in the final output. If the given file doesn’t have any WCS, then the default WCS (first file on the command-line with WCS) will be used in the output.

This option will mostly be used when the default file (first of the set of inputs) is not the one containing your desired WCS. But with this option, you can also use Arithmetic to rewrite/change the WCS of an existing FITS dataset from another file:

```
$ astarithmetic data.fits --wcsfile=other.fits -ofinal.fits
```

-W STR
--wcshdu STR
HDU/extension to read the WCS within the file given to --wcsfile. For more, see the description of --wcsfile.

--envseed
Use the environment for the random number generator settings in operators that need them (for example mknoise-sigma). This is very important for obtaining reproducible results, for more see Section 8.2.1.4 [Generating random numbers], page 464.

-O
--onedasimage
When final dataset to write as output only has one dimension, write it as a FITS image/array. By default, if the output is 1D, it will be written as a table, see above.

-s
--onedonstdout
When final dataset to write as output only has one dimension, print it on the standard output, not in a file. By default, if the output is 1D, it will be written as a table, see above.

-D
--dontdelete
Don’t delete the output file, or files given to the tofile or tofilefree operators, if they already exist. Instead append the desired datasets to the extensions that already exist in the respective file. Note it doesn’t matter if the final output file name is given with the --output option, or determined automatically.

Arithmetic treats this option differently from its default operation in other Gnuastro programs (see Section 4.1.2.1 [Input/Output options], page 157). If the output file exists, when other Gnuastro programs are called with
--dontdelete, they simply complain and abort. But when Arithmetic is called with --dontdelete, it will appended the dataset(s) to the existing extension(s) in the file.

Arithmetic accepts two kinds of input: images and numbers. Images are considered to be any of the inputs that is a file name of a recognized type (see Section 4.1.1.1 [Arguments], page 154) and has more than one element/pixel. Numbers on the command-line will be read into the smallest type (see Section 4.5 [Numeric data types], page 178) that can store them, so -2 will be read as a char type (which is signed on most systems and can thus keep negative values), 2500 will be read as an unsigned short (all positive numbers will be read as unsigned), while 3.1415926535897 will be read as a double and 3.14 will be read as a float. To force a number to be read as float, put a . after it (possibly followed by a zero for easier readability), or add an f after it. Hence while 5 will be read as an integer, 5., 5.0 or 5f will be added to the stack as float (see Section 6.2.1 [Reverse polish notation], page 272).

Unless otherwise stated (in Section 6.2.3 [Arithmetic operators], page 276), the operators can deal with numeric multiple data types (see Section 4.5 [Numeric data types], page 178). For example in "a.fits b.fits +", the image types can be long and float. In such cases, C’s internal type conversion will be used. The output type will be set to the higher-ranking type of the two inputs. Unsigned integer types have smaller ranking than their signed counterparts and floating point types have higher ranking than the integer types. So the internal C type conversions done in the example above are equivalent to this piece of C:

```c
size_t i;
long a[100];
float b[100], out[100];
for(i=0;i<100;++i) out[i]=a[i]+b[i];
```

Relying on the default C type conversion significantly speeds up the processing and also requires less RAM (when using very large images).

Some operators can only work on integer types (of any length, for example bitwise operators) while others only work on floating point types, (currently only the pow operator). In such cases, if the operand type(s) are different, an error will be printed. Arithmetic also comes with internal type conversion operators which you can use to convert the data into the appropriate type, see Section 6.2.3 [Arithmetic operators], page 276.

The hyphen (−) can be used both to specify options (see Section 4.1.1.2 [Options], page 154) and also to specify a negative number which might be necessary in your arithmetic. In order to enable you to do this, Arithmetic will first parse all the input strings and if the first character after a hyphen is a digit, then that hyphen is temporarily replaced by the vertical tab character which is not commonly used. The arguments are then parsed and these strings will not be specified as an option. Then the given arguments are parsed and any vertical tabs are replaced back with a hyphen so they can be read as negative numbers. Therefore, as long as the names of the files you want to work on, don’t start with a vertical tab followed by a digit, there is no problem. An important consequence of this implementation is that you should not write negative fractions like this: -.3, instead write them as -0.3.

Without any images, Arithmetic will act like a simple calculator and print the resulting output number on the standard output like the first example above. If you really want
such calculator operations on the command-line, AWK (GNU AWK is the most common implementation) is much faster, easier and much more powerful. For example, the numerical one-line example above can be done with the following command. In general AWK is a fantastic tool and GNU AWK has a wonderful manual (https://www.gnu.org/software/gawk/manual/). So if you often confront situations like this, or have to work with large text tables/catalogs, be sure to checkout AWK and simplify your life.

```
$ echo "" | awk '{print (10.32-3.84)^2.7}'
155.329
```

### 6.3 Convolve

On an image, convolution can be thought of as a process to blur or remove the contrast in an image. If you are already familiar with the concept and just want to run Convolve, you can jump to Section 6.3.4 [Convolution kernel], page 324, and Section 6.3.5 [Invoking Convolve], page 325, and skip the lengthy introduction on the basic definitions and concepts of convolution.

There are generally two methods to convolve an image. The first and more intuitive one is in the “spatial domain” or using the actual image pixel values, see Section 6.3.1 [Spatial domain convolution], page 306. The second method is when we manipulate the “frequency domain”, or work on the magnitudes of the different frequencies that constitute the image, see Section 6.3.2 [Frequency domain and Fourier operations], page 308. Understanding convolution in the spatial domain is more intuitive and thus recommended if you are just starting to learn about convolution. However, getting a good grasp of the frequency domain is a little more involved and needs some concentration and some mathematical proofs. However, its reward is a faster operation and more importantly a very fundamental understanding of this very important operation.

Convolution of an image will generally result in blurring the image because it mixes pixel values. In other words, if the image has sharp differences in neighboring pixel values\(^8\), those sharp differences will become smoother. This has very good consequences in detection of signal in noise for example. In an actual observed image, the variation in neighboring pixel values due to noise can be very high. But after convolution, those variations will decrease and we have a better hope in detecting the possible underlying signal. Another case where convolution is extensively used is in mock images and modeling in general, convolution can be used to simulate the effect of the atmosphere or the optical system on the mock profiles that we create, see Section 8.1.1.2 [Point spread function], page 442. Convolution is a very interesting and important topic in any form of signal analysis (including astronomical observations). So we have thoroughly\(^9\) explained the concepts behind it in the following sub-sections.

#### 6.3.1 Spatial domain convolution

The pixels in an input image represent different “spatial” positions, therefore when convolution is done only using the actual input pixel values, we name the process as being done

---

8 In astronomy, the only major time we confront such sharp borders in signal are cosmic rays. All other sources of signal in an image are already blurred by the atmosphere or the optics of the instrument.

9 A mathematician will certainly consider this explanation is incomplete and inaccurate. However this text is written for an understanding on the operations that are done on a real (not complex, discrete and noisy) astronomical image, not any general form of abstract function
Chapter 6: Data manipulation

in the “Spatial domain”. In particular this is in contrast to the “frequency domain” that we will discuss later in Section 6.3.2 [Frequency domain and Fourier operations], page 308. In the spatial domain (and in realistic situations where the image and the convolution kernel don’t extend to infinity), convolution is the process of changing the value of one pixel to the weighted average of all the pixels in its neighborhood.

The ‘neighborhood’ of each pixel (how many pixels in which direction) and the ‘weight’ function (how much each neighboring pixel should contribute depending on its position) are given through a second image which is known as a “kernel”\(^\text{10}\).

### 6.3.1.1 Convolution process

In convolution, the kernel specifies the weight and positions of the neighbors of each pixel. To find the convolved value of a pixel, the central pixel of the kernel is placed on that pixel. The values of each overlapping pixel in the kernel and image are multiplied by each other and summed for all the kernel pixels. To have one pixel in the center, the sides of the convolution kernel have to be an odd number. This process effectively mixes the pixel values of each pixel with its neighbors, resulting in a blurred image compared to the sharper input image.

Formally, convolution is one kind of linear ‘spatial filtering’ in image processing texts. If we assume that the kernel has \(2a + 1\) and \(2b + 1\) pixels on each side, the convolved value of a pixel placed at \(x\) and \(y\) \((C_{x,y})\) can be calculated from the neighboring pixel values in the input image \((I)\) and the kernel \((K)\) from

\[
C_{x,y} = \sum_{s=-a}^{a} \sum_{t=-b}^{b} K_{s,t} \times I_{x+s,y+t}.
\]

Any pixel coordinate that is outside of the image in the equation above will be considered to be zero. When the kernel is symmetric about its center the blurred image has the same orientation as the original image. However, if the kernel is not symmetric, the image will be affected in the opposite manner, this is a natural consequence of the definition of spatial filtering. In order to avoid this we can rotate the kernel about its center by 180 degrees so the convolved output can have the same original orientation. Technically speaking, only if the kernel is flipped the process is known Convolution. If it isn’t it is known as Correlation.

To be a weighted average, the sum of the weights (the pixels in the kernel) have to be unity. This will have the consequence that the convolved image of an object and unconvolved object will have the same brightness (see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394), which is natural, because convolution should not eat up the object photons, it only disperses them.

### 6.3.1.2 Edges in the spatial domain

In purely ‘linear’ spatial filtering (convolution), there are problems on the edges of the input image. Here we will explain the problem in the spatial domain. For a discussion of this problem from the frequency domain perspective, see Section 6.3.2.10 [Edges in the frequency domain], page 323. The problem originates from the fact that on the edges, in practice\(^\text{11}\),

\(^{10}\) Also known as filter, here we will use ‘kernel’.

\(^{11}\) Because we assumed the overlapping pixels outside the input image have a value of zero.
the sum of the weights we use on the actual image pixels is not unity. For example, as discussed above, a profile in the center of an image will have the same brightness before and after convolution. However, for partially imaged profile on the edge of the image, the brightness (sum of its pixel fluxes within the image, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394) will not be equal, some of the flux is going to be ‘eaten’ by the edges.

If you ran \$ make check on the source files of Gnuastro, you can see the this effect by comparing the convolve_frequency.fits with convolve_spatial.fits in the ./tests/ directory. In the spatial domain, by default, no assumption will be made about pixels outside of the image or any blank pixels in the image. The problem explained above will also occur on the sides of blank regions (see Section 6.1.3 [Blank pixels], page 262). The solution to this edge effect problem is only possible in the spatial domain. For pixels near the edge, we have to abandon the assumption that the sum of the kernel pixels is unity during the convolution process\(^\text{12}\). So taking \(W\) as the sum of the kernel pixels that overlapped with non-blank and in-image pixels, the equation in Section 6.3.1.1 [Convolution process], page 307, will become:

\[
C_{x,y} = \frac{\sum_{s=-a}^{a} \sum_{t=-b}^{b} K_{s,t} \times I_{x+s,y+t}}{W}.
\]

In this manner, objects which are near the edges of the image or blank pixels will also have the same brightness (within the image) before and after convolution. This correction is applied by default in Convolve when convolving in the spatial domain. To disable it, you can use the --noedgecorrection option. In the frequency domain, there is no way to avoid this loss of flux near the edges of the image, see Section 6.3.2.10 [Edges in the frequency domain], page 323, for an interpretation from the frequency domain perspective.

Note that the edge effect discussed here is different from the one in Section 8.1.2 [If convolving afterwards], page 446. In making mock images we want to simulate a real observation. In a real observation the images of the galaxies on the sides of the CCD are first blurred by the atmosphere and instrument, then imaged. So light from the parts of a galaxy which are immediately outside the CCD will affect the parts of the galaxy which are covered by the CCD. Therefore in modeling the observation, we have to convolve an image that is larger than the input image by exactly half of the convolution kernel. We can hence conclude that this correction for the edges is only useful when working on actual observed images (where we don’t have any more data on the edges) and not in modeling.

### 6.3.2 Frequency domain and Fourier operations

Getting a good grip on the frequency domain is usually not an easy job! So we have decided to give the issue a complete review here. Convolution in the frequency domain (see Section 6.3.2.6 [Convolution theorem], page 316) heavily relies on the concepts of Fourier transform (Section 6.3.2.4 [Fourier transform], page 314) and Fourier series (Section 6.3.2.3 [Fourier series], page 312) so we will be investigating these important operations first. It has become something of a cliché for people to say that the Fourier series “is a way to represent a (wave-like) function as the sum of simple sine waves” (from Wikipedia). However, sines

\(^{12}\) of course the sum of the kernel pixels still have to be unity in general.
themselves are abstract functions, so this statement really adds no extra layer of physical insight.

Before jumping head-first into the equations and proofs, we will begin with a historical background to see how the importance of frequencies actually roots in our ancient desire to see everything in terms of circles. A short review of how the complex plane should be interpreted is then given. Having paved the way with these two basics, we define the Fourier series and subsequently the Fourier transform. The final aim is to explain discrete Fourier transform, however some very important concepts need to be solidified first: The Dirac comb, convolution theorem and sampling theorem. So each of these topics are explained in their own separate sub-sub-section before going on to the discrete Fourier transform. Finally we revisit (after Section 6.3.1.2 [Edges in the spatial domain], page 307) the problem of convolution on the edges, but this time in the frequency domain. Understanding the sampling theorem and the discrete Fourier transform is very important in order to be able to pull out valuable science from the discrete image pixels. Therefore we have included the mathematical proofs and figures so you can have a clear understanding of these very important concepts.

6.3.2.1 Fourier series historical background

Ever since the ancient times, the circle has been (and still is) the simplest shape for abstract comprehension. All you need is a center point and a radius and you are done. All the points on a circle are at a fixed distance from the center. However, the moment you try to connect this elegantly simple and beautiful abstract construct (the circle) with the real world (for example compute its area or its circumference), things become really hard (ideally, impossible) because the irrational number $\pi$ gets involved.

The key to understanding the Fourier series (thus the Fourier transform and finally the Discrete Fourier Transform) is our ancient desire to express everything in terms of circles or the most exceptionally simple and elegant abstract human construct. Most people prefer to say the same thing in a more ahistorical manner: to break a function into sines and cosines. As the term “ancient” in the previous sentence implies, Jean-Baptiste Joseph Fourier (1768 – 1830 A.D.) was not the first person to do this. The main reason we know this process by his name today is that he came up with an ingenious method to find the necessary coefficients (radius of) and frequencies (“speed” of rotation on) the circles for any generic (integrable) function.
Figure 6.1: Epicycles and the Fourier series. Left: A demonstration of Mercury’s epicycles relative to the “center of the world” by Qutb al-Din al-Shirazi (1236 – 1311 A.D.) retrieved from Wikipedia (https://commons.wikimedia.org/wiki/File:Ghotb2.jpg). Middle (https://commons.wikimedia.org/wiki/File:Fourier_series_square_wave_circles_animation.gif) and Right: How adding more epicycles (or terms in the Fourier series) will approximate functions. The right (https://commons.wikimedia.org/wiki/File:Fourier_series_sawtooth_wave_circles_animation.gif) animation is also available.

Like most aspects of mathematics, this process of interpreting everything in terms of circles, began for astronomical purposes. When astronomers noticed that the orbit of Mars and other outer planets, did not appear to be a simple circle (as everything should have been in the heavens). At some point during their orbit, the revolution of these planets would become slower, stop, go back a little (in what is known as the retrograde motion) and then continue going forward again.

The correction proposed by Ptolemy (90 – 168 A.D.) was the most agreed upon. He put the planets on Epicycles or circles whose center itself rotates on a circle whose center is the earth. Eventually, as observations became more and more precise, it was necessary to add more and more epicycles in order to explain the complex motions of the planets\textsuperscript{13}. Figure 6.1(Left) shows an example depiction of the epicycles of Mercury in the late 13th century.

Of course we now know that if they had abdicated the Earth from its throne in the center of the heavens and allowed the Sun to take its place, everything would become much simpler and true. But there wasn’t enough observational evidence for changing the “professional consensus” of the time to this radical view suggested by a small minority\textsuperscript{14}. So the pre-

\textsuperscript{13} See the Wikipedia page on “Deferent and epicycle” for a more complete historical review.

\textsuperscript{14} Aristarchus of Samos (310 – 230 B.C.) appears to be one of the first people to suggest the Sun being in the center of the universe. This approach to science (that the standard model is defined by consensus) and the fact that this consensus might be completely wrong still applies equally well to our models of particle physics and cosmology today.
Galilean astronomers chose to keep Earth in the center and find a correction to the models (while keeping the heavens a purely “circular” order).

The main reason we are giving this historical background which might appear off topic is to give historical evidence that while such “approximations” do work and are very useful for pragmatic reasons (like measuring the calendar from the movement of astronomical bodies). They offer no physical insight. The astronomers who were involved with the Ptolemaic world view had to add a huge number of epicycles during the centuries after Ptolemy in order to explain more accurate observations. Finally the death knell of this world-view was Galileo’s observations with his new instrument (the telescope). So the physical insight, which is what Astronomers and Physicists are interested in (as opposed to Mathematicians and Engineers who just like proving and optimizing or calculating!) comes from being creative and not limiting our selves to such approximations. Even when they work.

6.3.2.2 Circles and the complex plane

Before going onto the derivation, it is also useful to review how the complex numbers and their plane relate to the circles we talked about above. The two schematics in the middle and right of Figure 6.1 show how a 1D function of time can be made using the 2D real and imaginary surface. Seeing the animation in Wikipedia will really help in understanding this important concept. At each point in time, we take the vertical coordinate of the point and use it to find the value of the function at that point in time. Figure 6.2 shows this relation with the axes marked.

Leonhard Euler\(^{15}\) (1707 – 1783 A.D.) showed that the complex exponential \( (e^{iv} \text{ where } v \text{ is real}) \) is periodic and can be written as: \( e^{iv} = \cos v + iv \sin v \). Therefore \( e^{iv+2\pi} = e^{iv} \). Later, Caspar Wessel (mathematician and cartographer 1745 – 1818 A.D.) showed how complex numbers can be displayed as vectors on a plane. Euler’s identity might seem counter intuitive at first, so we will try to explain it geometrically (for deeper physical insight). On the real-imaginary 2D plane (like the left hand plot in each box of Figure 6.2), multiplying a number by \( i \) can be interpreted as rotating the point by 90 degrees (for example the value 3 on the real axis becomes \( 3i \) on the imaginary axis). On the other hand, \( e \equiv \lim_{n \to \infty} (1 + \frac{1}{n})^n \), therefore, defining \( m \equiv nu \), we get:

\[
e^u = \lim_{n \to \infty} \left(1 + \frac{1}{n}\right)^{nu} = \lim_{n \to \infty} \left(1 + \frac{u}{nu}\right)^{nu} = \lim_{m \to \infty} \left(1 + \frac{u}{m}\right)^m
\]

Taking \( u \equiv iv \) the result can be written as a generic complex number (a function of \( v \)):

\[
e^{iv} = \lim_{m \to \infty} \left(1 + \frac{v}{m}\right)^m = a(v) + ib(v)
\]

For \( v = \pi \), a nice geometric animation of going to the limit can be seen on Wikipedia (https://commons.wikimedia.org/wiki/File:ExpIPi.gif). We see that \( \lim_{m \to \infty} a(\pi) = -1 \), while \( \lim_{m \to \infty} b(\pi) = 0 \), which gives the famous \( e^{i\pi} = -1 \) equation. The final value is

---

\(^{15}\) Other forms of this equation were known before Euler. For example in 1707 A.D. (the year of Euler’s birth) Abraham de Moivre (1667 – 1754 A.D.) showed that \( (\cos x + i \sin x)^n = \cos(nx) + i \sin(nx) \). In 1714 A.D., Roger Cotes (1682 – 1716 A.D. a colleague of Newton who proofread the second edition of Principia) showed that: \( ix = \ln(\cos x + i \sin x) \).
the real number $-1$, however the distance of the polygon points traversed as $m \to \infty$ is half the circumference of a circle or $\pi$, showing how $v$ in the equation above can be interpreted as an angle in units of radians and therefore how $a(v) = \cos(v)$ and $b(v) = \sin(v)$.

Since $e^{iv}$ is periodic (let’s assume with a period of $T$), it is more clear to write it as $v \equiv \frac{2\pi n}{T}$ (where $n$ is an integer), so $e^{iv} = e^{i\frac{2\pi n}{T}t}$. The advantage of this notation is that the period ($T$) is clearly visible and the frequency ($\frac{2\pi n}{T}$, in units of 1/cycle) is defined through the integer $n$. In this notation, $t$ is in units of “cycle”s.

As we see from the examples in Figure 6.1 and Figure 6.2, for each constituting frequency, we need a respective ‘magnitude’ or the radius of the circle in order to accurately approximate the desired 1D function. The concepts of “period” and “frequency” are relatively easy to grasp when using temporal units like time because this is how we define them in everyday life. However, in an image (astronomical data), we are dealing with spatial units like distance. Therefore, by one “period” we mean the distance at which the signal is identical and frequency is defined as the inverse of that spatial “period”. The complex circle of Figure 6.2 can be thought of the Moon rotating about Earth which is rotating around the Sun; so the “Real (signal)” axis shows the Moon’s position as seen by a distant observer on the Sun as time goes by. Because of the scalar (not having any direction or vector) nature of time, Figure 6.2 is easier to understand in units of time. When thinking about spatial units, mentally replace the “Time (sec)” axis with “Distance (meters)”. Because length has direction and is a vector, visualizing the rotation of the imaginary circle and the advance along the “Distance (meters)” axis is not as simple as temporal units like time.

Figure 6.2: Relation between the real (signal), imaginary ($i \equiv \sqrt{-1}$) and time axes at two snapshots of time.

### 6.3.2.3 Fourier series

In astronomical images, our variable (brightness, or number of photo-electrons, or signal to be more generic) is recorded over the 2D spatial surface of a camera pixel. However to make things easier to understand, here we will assume that the signal is recorded in 1D (assume one row of the 2D image pixels). Also for this section and the next (Section 6.3.2.4 [Fourier transform], page 314) we will be talking about the signal before it is digitized or pixelated. Let’s assume that we have the continuous function $f(l)$ which is integrable in the interval $[l_0, l_0 + L]$ (always true in practical cases like images). Take $l_0$ as the position of the first pixel in the assumed row of the image and $L$ as the width of the image along that row. The units of $l_0$ and $L$ can be in any spatial units (for example meters) or an angular unit (like radians) multiplied by a fixed distance which is more common.
To approximate $f(l)$ over this interval, we need to find a set of frequencies and their corresponding ‘magnitude’s (see Section 6.3.2.2 [Circles and the complex plane], page 311). Therefore our aim is to show $f(l)$ as the following sum of periodic functions:

$$f(l) = \sum_{n=-\infty}^{\infty} c_n e^{i \frac{2\pi n}{L} l}$$

Note that the different frequencies ($2\pi n/L$, in units of cycles per meters for example) are not arbitrary. They are all integer multiples of the fundamental frequency of $\omega_0 = 2\pi/L$. Recall that $L$ was the length of the signal we want to model. Therefore, we see that the smallest possible frequency (or the frequency resolution) in the end, depends on the length we observed the signal or $L$. In the case of each dimension on an image, this is the size of the image in the respective dimension. The frequencies have been defined in this “harmonic” fashion to insure that the final sum is periodic outside of the $[l_0, l_0 + L]$ interval too. At this point, you might be thinking that the sky is not periodic with the same period as my camera’s view angle. You are absolutely right! The important thing is that since your camera’s observed region is the only region we are “observing” and will be using, the rest of the sky is irrelevant; so we can safely assume the sky is periodic outside of it. However, this working assumption will haunt us later in Section 6.3.2.10 [Edges in the frequency domain], page 323.

The frequencies are thus determined by definition. So all we need to do is to find the coefficients ($c_n$), or magnitudes, or radii of the circles for each frequency which is identified with the integer $n$. Fourier’s approach was to multiply both sides with a fixed term:

$$f(l)e^{-i \frac{2\pi m}{L} l} = \sum_{n=-\infty}^{\infty} c_n e^{i \frac{2\pi (n-m)}{L} l}$$

where $m > 0$. We can then integrate both sides over the observation period:

$$\int_{l_0}^{l_0+L} f(l)e^{-i \frac{2\pi m}{L} l} dl = \int_{l_0}^{l_0+L} \sum_{n=-\infty}^{\infty} c_n e^{i \frac{2\pi (n-m)}{L} l} dl = \sum_{n=-\infty}^{\infty} c_n \int_{l_0}^{l_0+L} e^{i \frac{2\pi (n-m)}{L} l} dl$$

Both $n$ and $m$ are positive integers. Also, we know that a complex exponential is periodic so after one period ($L$) it comes back to its starting point. Therefore $\int_{l_0}^{l_0+L} e^{i 2\pi k/L} dl = 0$ for any $k > 0$. However, when $k = 0$, this integral becomes: $\int_{l_0}^{l_0+T} e^{0} dt = \int_{l_0}^{l_0+T} dt = T$. Hence since the integral will be zero for all $n \neq m$, we get:

$$\sum_{n=-\infty}^{\infty} c_n \int_{l_0}^{l_0+T} e^{i \frac{2\pi (n-m)}{L} l} dl = Lc_m$$

The origin of the axis is fundamentally an arbitrary position. So let’s set it to the start of the image such that $l_0 = 0$. So we can find the “magnitude” of the frequency $2\pi m/L$ within $f(l)$ through the relation:

---

16 We could have assumed $m < 0$ and set the exponential to positive, but this is more clear.
\[ c_m = \frac{1}{L} \int_0^L f(l) e^{-i\frac{2\pi m}{L} l} \, dl \]

### 6.3.2.4 Fourier transform

In Section 6.3.2.3 [Fourier series], page 312, we had to assume that the function is periodic outside of the desired interval with a period of \( L \). Therefore, assuming that \( L \to \infty \) will allow us to work with any function. However, with this approximation, the fundamental frequency \( (\omega_0) \) or the frequency resolution that we discussed in Section 6.3.2.3 [Fourier series], page 312, will tend to zero: \( \omega_0 \to 0 \). In the equation to find \( c_m \), every \( m \) represented a frequency (multiple of \( \omega_0 \)) and the integration on \( l \) removes the dependence of the right side of the equation on \( l \), making it only a function of \( m \) or frequency. Let’s define the following two variables:

\[ \omega \equiv m \omega_0 = \frac{2\pi m}{L} \]

\[ F(\omega) \equiv Lc_m \]

The equation to find the coefficients of each frequency in Section 6.3.2.3 [Fourier series], page 312, thus becomes:

\[ F(\omega) = \int_{-\infty}^{\infty} f(l) e^{-i\omega l} \, dl. \]

The function \( F(\omega) \) is thus the Fourier transform of \( f(l) \) in the frequency domain. So through this transformation, we can find (analyze) the magnitudes of the constituting frequencies or the value in the frequency space\(^\text{17} \) of our spatial input function. The great thing is that we can also do the reverse and later synthesize the input function from its Fourier transform. Let’s do it: with the approximations above, multiply the right side of the definition of the Fourier Series (Section 6.3.2.3 [Fourier series], page 312) with \( 1 = L/L = (\omega_0 L)/(2\pi) \):

\[ f(l) = \frac{1}{2\pi} \sum_{n=-\infty}^{\infty} Lc_n e^{\frac{2\pi in}{\omega_0} l} = \frac{1}{2\pi} \sum_{n=-\infty}^{\infty} F(\omega) e^{i\omega l} \Delta \omega \]

To find the right most side of this equation, we renamed \( \omega_0 \) as \( \Delta \omega \) because it was our resolution, \( 2\pi n/L \) was written as \( \omega \) and finally, \( Lc_n \) was written as \( F(\omega) \) as we defined above. Now, as \( L \to \infty, \Delta \omega \to 0 \) so we can write:

\[ f(l) = \frac{1}{2\pi} \int_{-\infty}^{\infty} F(\omega) e^{i\omega l} d\omega \]

\(^\text{17}\) As we discussed before, this ‘magnitude’ can be interpreted as the radius of the circle rotating at this frequency in the epicyclic interpretation of the Fourier series, see Figure 6.1 and Figure 6.2.
Together, these two equations provide us with a very powerful set of tools that we can use to process (analyze) and recreate (synthesize) the input signal. Through the first equation, we can break up our input function into its constituent frequencies and analyze it, hence it is also known as analysis. Using the second equation, we can synthesize or make the input function from the known frequencies and their magnitudes. Thus it is known as synthesis. Here, we symbolize the Fourier transform (analysis) and its inverse (synthesis) of a function $f(l)$ and its Fourier Transform $F(\omega)$ as $\mathcal{F}[f]$ and $\mathcal{F}^{-1}[F]$.

### 6.3.2.5 Dirac delta and comb

The Dirac $\delta$ (delta) function (also known as an impulse) is the way that we convert a continuous function into a discrete one. It is defined to satisfy the following integral:

$$\int_{-\infty}^{\infty} \delta(l) dl = 1$$

When integrated with another function, it gives that function’s value at $l = 0$:

$$\int_{-\infty}^{\infty} f(l) \delta(l) dt = f(0)$$

An impulse positioned at another point (say $l_0$) is written as $\delta(l - l_0)$:

$$\int_{-\infty}^{\infty} f(l) \delta(l - l_0) dt = f(l_0)$$

The Dirac $\delta$ function also operates similarly if we use summations instead of integrals. The Fourier transform of the delta function is:

$$\mathcal{F}[\delta(l)] = \int_{-\infty}^{\infty} \delta(l) e^{-i\omega l} dl = e^{-i\omega 0} = 1$$

$$\mathcal{F}[\delta(l - l_0)] = \int_{-\infty}^{\infty} \delta(l - l_0) e^{-i\omega l} dl = e^{-i\omega l_0}$$

From the definition of the Dirac $\delta$ we can also define a Dirac comb (III$_P$) or an impulse train with infinite impulses separated by $P$:

$$\text{III}_P(l) \equiv \sum_{k=-\infty}^{\infty} \delta(l - kP)$$

$P$ is chosen to represent “pixel width” later in Section 6.3.2.7 [Sampling theorem], page 318. Therefore the Dirac comb is periodic with a period of $P$. We have intentionally used a different name for the period of the Dirac comb compared to the input signal’s length of observation that we showed with $L$ in Section 6.3.2.3 [Fourier series], page 312. This
difference is highlighted here to avoid confusion later when these two periods are needed together in Section 6.3.2.8 [Discrete Fourier transform], page 320. The Fourier transform of the Dirac comb will be necessary in Section 6.3.2.7 [Sampling theorem], page 318, so let’s derive it. By its definition, it is periodic, with a period of $P$, so the Fourier coefficients of its Fourier Series (Section 6.3.2.3 [Fourier series], page 312) can be calculated within one period:

$$III_P = \sum_{n=-\infty}^{\infty} c_n e^{\frac{2\pi in}{P}}$$

We can now find the $c_n$ from Section 6.3.2.3 [Fourier series], page 312:

$$c_n = \frac{1}{P} \int_{-P/2}^{P/2} \delta(l) e^{-\frac{2\pi in}{P} l} = \frac{1}{P} \rightarrow III_P = \frac{1}{P} \sum_{n=-\infty}^{\infty} e^{\frac{2\pi in}{P}}$$

So we can write the Fourier transform of the Dirac comb as:

$$\mathcal{F}[III_P] = \int_{-\infty}^{\infty} III_P e^{-i\omega l} dl = \frac{1}{P} \sum_{n=-\infty}^{\infty} \int_{-\infty}^{\infty} e^{-i(\omega - \frac{2\pi n}{P}) l} dl = \frac{1}{P} \sum_{n=-\infty}^{\infty} \delta \left( \omega - \frac{2\pi n}{P} \right)$$

In the last step, we used the fact that the complex exponential is a periodic function, that $n$ is an integer and that as we defined in Section 6.3.2.4 [Fourier transform], page 314, $\omega \equiv m\omega_0$, where $m$ was an integer. The integral will be zero for any $\omega$ that is not equal to $2\pi n/P$, a more complete explanation can be seen in Section 6.3.2.3 [Fourier series], page 312. Therefore, while in the spatial domain the impulses had spacing of $P$ (meters for example), in the frequency space, the spacing between the different impulses are $2\pi/P$ cycles per meters.

### 6.3.2.6 Convolution theorem

The convolution (shown with the $*$ operator) of the two functions $f(l)$ and $h(l)$ is defined as:

$$c(l) \equiv [f * h](l) = \int_{-\infty}^{\infty} f(\tau) h(l - \tau) d\tau$$

See Section 6.3.1.1 [Convolution process], page 307, for a more detailed physical (pixel based) interpretation of this definition. The Fourier transform of convolution ($C(\omega)$) can be written as:

$$C(\omega) = \int_{-\infty}^{\infty} [f * h](l) e^{-i\omega l} dl = \int_{-\infty}^{\infty} f(\tau) \left[ \int_{-\infty}^{\infty} h(l - \tau) e^{-i\omega l} dl \right] d\tau$$

To solve the inner integral, let’s define $s \equiv l - \tau$, so that $ds = dl$ and $l = s + \tau$ then the inner integral becomes:
\[ \int_{-\infty}^{\infty} h(l - \tau)e^{-i\omega l}dl = \int_{-\infty}^{\infty} h(s)e^{-i\omega (s+\tau)}ds = e^{-i\omega \tau}\int_{-\infty}^{\infty} h(s)e^{-i\omega s}ds = H(\omega)e^{-i\omega \tau} \]

where \( H(\omega) \) is the Fourier transform of \( h(l) \). Substituting this result for the inner integral above, we get:

\[ C(\omega) = H(\omega)\int_{-\infty}^{\infty} f(\tau)e^{-i\omega \tau}d\tau = H(\omega)F(\omega) = F(\omega)H(\omega) \]

where \( F(\omega) \) is the Fourier transform of \( f(l) \). So multiplying the Fourier transform of two functions individually, we get the Fourier transform of their convolution. The convolution theorem also proves a relation between the convolutions in the frequency space. Let’s define:

\[ D(\omega) \equiv F(\omega) \ast H(\omega) \]

Applying the inverse Fourier Transform or synthesis equation (Section 6.3.2.4 [Fourier transform], page 314) to both sides and following the same steps above, we get:

\[ d(l) = f(l)h(l) \]

Where \( d(l) \) is the inverse Fourier transform of \( D(\omega) \). We can therefore re-write the two equations above formally as the convolution theorem:

\[ \mathcal{F}[f \ast h] = \mathcal{F}[f] \mathcal{F}[h] \]

\[ \mathcal{F}[fh] = \mathcal{F}[f] \ast \mathcal{F}[h] \]

Besides its usefulness in blurring an image by convolving it with a given kernel, the convolution theorem also enables us to do another very useful operation in data analysis: to match the blur (or PSF) between two images taken with different telescopes/cameras or under different atmospheric conditions. This process is also known as de-convolution. Let’s take \( f(l) \) as the image with a narrower PSF (less blurry) and \( c(l) \) as the image with a wider PSF which appears more blurred. Also let’s take \( h(l) \) to represent the kernel that should be convolved with the sharper image to create the more blurry image. Above, we proved the relation between these three images through the convolution theorem. But there, we assumed that \( f(l) \) and \( h(l) \) are known (given) and the convolved image is desired.

In de-convolution, we have \( f(l) \) – the sharper image – and \( f \ast h(l) \) – the more blurry image – and we want to find the kernel \( h(l) \). The solution is a direct result of the convolution theorem:
\[ F[h] = \frac{F[f * h]}{F[f]} \quad \text{or} \quad h(l) = F^{-1} \left[ \frac{F[f * h]}{F[f]} \right] \]

While this works really nice, it has two problems:

- If \( F[f] \) has any zero values, then the inverse Fourier transform will not be a number!
- If there is significant noise in the image, then the high frequencies of the noise are going to significantly reduce the quality of the final result.

A standard solution to both these problems is the Weiner de-convolution algorithm\(^{18}\).

### 6.3.2.7 Sampling theorem

Our mathematical functions are continuous, however, our data collecting and measuring tools are discrete. Here we want to give a mathematical formulation for digitizing the continuous mathematical functions so that later, we can retrieve the continuous function from the digitized recorded input. Assuming that we have a continuous function \( f(l) \), then we can define \( f_s(l) \) as the ‘sampled’ \( f(l) \) through the Dirac comb (see Section 6.3.2.5 [Dirac delta and comb], page 315):

\[ f_s(l) = f(l) \text{III}_P = \sum_{n=-\infty}^{\infty} f(l) \delta(l - nP) \]

The discrete data-element \( f_k \) (for example, a pixel in an image), where \( k \) is an integer, can thus be represented as:

\[ f_k = \int_{-\infty}^{\infty} f_s(l) dl = \int_{-\infty}^{\infty} f(l) \delta(l - kP) dt = f(kP) \]

Note that in practice, our discrete data points are not found in this fashion. Each detector pixel (in an image for example) has an area and averages the signal it receives over that area, not a mathematical point as the Dirac \( \delta \) function defines. However, as long as the variation in the signal over one detector pixel is not significant, this can be a good approximation. Having put this issue to the side, we can now try to find the relation between the Fourier transforms of the un-sampled \( f(l) \) and the sampled \( f_s(l) \). For a more clear notation, let’s define:

\[ F_s(\omega) \equiv F[f_s] \]

\[ D(\omega) \equiv F[\text{III}_P] \]

Then using the Convolution theorem (see Section 6.3.2.6 [Convolution theorem], page 316), \( F_s(\omega) \) can be written as:

\(^{18}\) https://en.wikipedia.org/wiki/Wiener_deconvolution
Finally, from the definition of convolution and the Fourier transform of the Dirac comb (see Section 6.3.2.5 [Dirac delta and comb], page 315), we get:

\[ F_s(\omega) = \mathcal{F}[f(l)\Pi P] = F(\omega) * D(\omega) \]

\[
F_s(\omega) = \int_{-\infty}^{\infty} F(\omega) D(\omega - \mu) d\mu \\
= \frac{1}{P} \sum_{n=-\infty}^{\infty} \int_{-\infty}^{\infty} F(\omega) \delta \left( \omega - \mu - \frac{2\pi n}{P} \right) d\mu \\
= \frac{1}{P} \sum_{n=-\infty}^{\infty} F \left( \omega - \frac{2\pi n}{P} \right).
\]

\( F(\omega) \) was only a simple function, see Figure 6.3(left). However, from the sampled Fourier transform function we see that \( F_s(\omega) \) is the superposition of infinite copies of \( F(\omega) \) that have been shifted, see Figure 6.3(right). From the equation, it is clear that the shift in each copy is \( 2\pi/P \).

---

**Figure 6.3:** Sampling causes infinite repetition in the frequency domain. FT is an abbreviation for ‘Fourier transform’. \( \omega_m \) represents the maximum frequency present in the input. \( F(\omega) \) is only symmetric on both sides of 0 when the input is real (not complex). In general \( F(\omega) \) is complex and thus cannot be simply plotted like this. Here we have assumed a real Gaussian \( f(t) \) which has produced a Gaussian \( F(\omega) \).

The input \( f(l) \) can have any distribution of frequencies in it. In the example of Figure 6.3(left), the input consisted of a range of frequencies equal to \( \Delta \omega = 2\omega_m \). Fortunately as Figure 6.3(right) shows, the assumed pixel size \( (P) \) we used to sample this hypothetical function was such that \( 2\pi/P > \Delta \omega \). The consequence is that each copy of \( F(\omega) \) has become completely separate from the surrounding copies. Such a digitized (sampled) data set is thus called *over-sampled*. When \( 2\pi/P = \Delta \omega \), \( P \) is just small enough to finely separate even the largest frequencies in the input signal and thus it is known as *critically-sampled*. Finally if \( 2\pi/P < \Delta \omega \) we are dealing with an *under-sampled* data set. In an under-sampled data set, the separate copies of \( F(\omega) \) are going to overlap and this will deprive us of recovering high constituent frequencies of \( f(l) \). The effects of under-sampling in an image with high rates of change (for example a brick wall imaged from a distance) can clearly be visually seen and is known as *aliasing*. 
When the input \( f(l) \) is composed of a finite range of frequencies, \( f(l) \) is known as a \textit{band-limited} function. The example in Figure 6.3(left) was a nice demonstration of such a case: for all \( \omega < -\omega_m \) or \( \omega > \omega_m \), we have \( F(\omega) = 0 \). Therefore, when the input function is band-limited and our detector’s pixels are placed such that we have critically (or over-) sampled it, then we can exactly reproduce the continuous \( f(l) \) from the discrete or digitized samples. To do that, we just have to isolate one copy of \( F(\omega) \) from the infinite copies and take its inverse Fourier transform.

This ability to exactly reproduce the continuous input from the sampled or digitized data leads us to the \textit{sampling theorem} which connects the inherent property of the continuous signal (its maximum frequency) to that of the detector (the spacing between its pixels). The sampling theorem states that the full (continuous) signal can be recovered when the pixel size \( (P) \) and the maximum constituent frequency in the signal \( (\omega_m) \) have the following relation\(^{19}:\)

\[
\frac{2\pi}{P} > 2\omega_m
\]

This relation was first formulated by Harry Nyquist (1889 – 1976 A.D.) in 1928 and formally proved in 1949 by Claude E. Shannon (1916 – 2001 A.D.) in what is now known as the Nyquist-Shannon sampling theorem. In signal processing, the signal is produced (synthesized) by a transmitter and is received and de-coded (analyzed) by a receiver. Therefore producing a band-limited signal is necessary.

In astronomy, we do not produce the shapes of our targets, we are only observers. Galaxies can have any shape and size, therefore ideally, our signal is not band-limited. However, since we are always confined to observing through an aperture, the aperture will cause a point source (for which \( \omega_m = \infty \)) to be spread over several pixels. This spread is quantitatively known as the point spread function or PSF. This spread does blur the image which is undesirable; however, for this analysis it produces the positive outcome that there will be a finite \( \omega_m \). Though we should caution that any detector will have noise which will add lots of very high frequency (ideally infinite) changes between the pixels. However, the coefficients of those noise frequencies are usually exceedingly small.

\subsection{Discrete Fourier transform}

As we have stated several times so far, the input image is a digitized, pixelated or discrete array of values \((f_s(l))\), see Section 6.3.2.7 [Sampling theorem], page 318). The input is not a continuous function. Also, all our numerical calculations can only be done on a sampled, or discrete Fourier transform. Note that \( F_s(\omega) \) is not discrete, it is continuous. One way would be to find the analytic \( F_s(\omega) \), then sample it at any desired “freq-pixel”\(^{20} \) spacing. However, this process would involve two steps of operations and computers in particular are not too good at analytic operations for the first step. So here, we will derive a method to directly find the ‘freq-pixel’ated \( F_s(\omega) \) from the pixelated \( f_s(l) \). Let’s start with the definition of the Fourier transform (see Section 6.3.2.4 [Fourier transform], page 314):

\(^{19}\) This equation is also shown in some places without the \( 2\pi \). Whether \( 2\pi \) is included or not depends on how you define the frequency

\(^{20}\) We are using the made-up word “freq-pixel” so they are not confused with spatial domain “pixels”. 
\[ F_s(\omega) = \int_{-\infty}^{\infty} f_s(l)e^{-i\omega l} dl \]

From the definition of \( f_s(\omega) \) (using \( x \) instead of \( n \)) we get:

\[ F_s(\omega) = \sum_{x=-\infty}^{\infty} \int_{-\infty}^{\infty} f(l)\delta(l - xP)e^{-i\omega l} dl \]

\[ = \sum_{x=-\infty}^{\infty} f_x e^{-i\omega xP} \]

Where \( f_x \) is the value of \( f(l) \) on the point \( x \) or the value of the \( x \)th pixel. As shown in Section 6.3.2.7 [Sampling theorem], page 318, this function is infinitely periodic with a period of \( 2\pi/P \). So all we need is the values within one period: \( 0 < \omega < 2\pi/P \), see Figure 6.3. We want \( X \) samples within this interval, so the frequency difference between each frequency sample or freq-pixel is \( 1/XP \). Hence we will evaluate the equation above on the points at:

\[ \omega = \frac{u}{XP} \quad u = 0, 1, 2, ..., X - 1 \]

Therefore the value of the freq-pixel \( u \) in the frequency domain is:

\[ F_u = \sum_{x=0}^{X-1} f_x e^{-i\frac{ux}{X}} \]

Therefore, we see that for each freq-pixel in the frequency domain, we are going to need all the pixels in the spatial domain\(^{21}\). If the input (spatial) pixel row is also \( X \) pixels wide, then we can exactly recover the \( x \)th pixel with the following summation:

\[ f_x = \frac{1}{X} \sum_{u=0}^{X-1} F_u e^{i\frac{ux}{X}} \]

When the input pixel row (we are still only working on 1D data) has \( X \) pixels, then it is \( L = XP \) spatial units wide. \( L \), or the length of the input data was defined in Section 6.3.2.3 [Fourier series], page 312, and \( P \) or the space between the pixels in the input was defined in Section 6.3.2.5 [Dirac delta and comb], page 315. As we saw in Section 6.3.2.7 [Sampling theorem], page 318, the input (spatial) pixel spacing (\( P \)) specifies the range of frequencies that can be studied and in Section 6.3.2.3 [Fourier series], page 312, we saw that the length of the (spatial) input, (\( L \)) determines the resolution (or size of the freq-pixels) in our discrete Fourier transformed image. Both result from the fact that the frequency domain is the inverse of the spatial domain.

\(^{21}\) So even if one pixel is a blank pixel (see Section 6.1.3 [Blank pixels], page 262), all the pixels in the frequency domain will also be blank.
6.3.2.9 Fourier operations in two dimensions

Once all the relations in the previous sections have been clearly understood in one dimension, it is very easy to generalize them to two or even more dimensions since each dimension is by definition independent. Previously we defined $l$ as the continuous variable in 1D and the inverse of the period in its direction to be $\omega$. Let’s show the second spatial direction with $m$ the inverse of the period in the second dimension with $\nu$. The Fourier transform in 2D (see Section 6.3.2.4 [Fourier transform], page 314) can be written as:

$$F(\omega, \nu) = \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} f(l, m) e^{-i(\omega l + \nu m)} dl$$

$$f(l, m) = \int_{-\infty}^{\infty} \int_{-\infty}^{\infty} F(\omega, \nu) e^{i(\omega l + \nu m)} dl$$

The 2D Dirac $\delta(l, m)$ is non-zero only when $l = m = 0$. The 2D Dirac comb (or Dirac brush! See Section 6.3.2.5 [Dirac delta and comb], page 315) can be written in units of the 2D Dirac $\delta$. For most image detectors, the sides of a pixel are equal in both dimensions. So $P$ remains unchanged, if a specific device is used which has non-square pixels, then for each dimension a different value should be used.

$$\Pi_P(l, m) \equiv \sum_{j=-\infty}^{\infty} \sum_{k=-\infty}^{\infty} \delta(l - jP, m - kP)$$

The Two dimensional Sampling theorem (see Section 6.3.2.7 [Sampling theorem], page 318) is thus very easily derived as before since the frequencies in each dimension are independent. Let’s take $\nu_m$ as the maximum frequency along the second dimension. Therefore the two dimensional sampling theorem says that a 2D band-limited function can be recovered when the following conditions hold:\footnote{If the pixels are not a square, then each dimension has to use the respective pixel size, but since most detectors have square pixels, we assume so here too}:

$$\frac{2\pi}{P} > 2\omega_m \quad \text{and} \quad \frac{2\pi}{P} > 2\nu_m$$

Finally, let’s represent the pixel counter on the second dimension in the spatial and frequency domains with $y$ and $v$ respectively. Also let’s assume that the input image has $Y$ pixels on the second dimension. Then the two dimensional discrete Fourier transform and its inverse (see Section 6.3.2.8 [Discrete Fourier transform], page 320) can be written as:

$$F_{u,v} = \sum_{x=0}^{X-1} \sum_{y=0}^{Y-1} f_{x,y} e^{-i\left(\frac{2\pi}{X}x + \frac{2\pi}{Y}y\right)}$$

$$f_{x,y} = \frac{1}{XY} \sum_{u=0}^{X-1} \sum_{v=0}^{Y-1} F_{u,v} e^{i\left(\frac{2\pi}{X}x + \frac{2\pi}{Y}y\right)}$$
6.3.2.10 Edges in the frequency domain

With a good grasp of the frequency domain, we can revisit the problem of convolution on the image edges, see Section 6.3.1.2 [Edges in the spatial domain], page 307. When we apply the convolution theorem (see Section 6.3.2.6 [Convolution theorem], page 316) to convolve an image, we first take the discrete Fourier transforms (DFT, Section 6.3.2.8 [Discrete Fourier transform], page 320) of both the input image and the kernel, then we multiply them with each other and then take the inverse DFT to construct the convolved image. Of course, in order to multiply them with each other in the frequency domain, the two images have to be the same size, so let’s assume that we pad the kernel (it is usually smaller than the input image) with zero valued pixels in both dimensions so it becomes the same size as the input image before the DFT.

Having multiplied the two DFTs, we now apply the inverse DFT which is where the problem is usually created. If the DFT of the kernel only had values of 1 (unrealistic condition!) then there would be no problem and the inverse DFT of the multiplication would be identical with the input. However in real situations, the kernel’s DFT has a maximum of 1 (because the sum of the kernel has to be one, see Section 6.3.1.1 [Convolution process], page 307) and decreases something like the hypothetical profile of Figure 6.3. So when multiplied with the input image’s DFT, the coefficients or magnitudes (see Section 6.3.2.2 [Circles and the complex plane], page 311) of the smallest frequency (or the sum of the input image pixels) remains unchanged, while the magnitudes of the higher frequencies are significantly reduced.

As we saw in Section 6.3.2.7 [Sampling theorem], page 318, the Fourier transform of a discrete input will be infinitely repeated. In the final inverse DFT step, the input is in the frequency domain (the multiplied DFT of the input image and the kernel DFT). So the result (our output convolved image) will be infinitely repeated in the spatial domain. In order to accurately reconstruct the input image, we need all the frequencies with the correct magnitudes. However, when the magnitudes of higher frequencies are decreased, longer periods (shorter frequencies) will dominate in the reconstructed pixel values. Therefore, when constructing a pixel on the edge of the image, the newly empowered longer periods will look beyond the input image edges and will find the repeated input image there. So if you convolve an image in this fashion using the convolution theorem, when a bright object exists on one edge of the image, its blurred wings will be present on the other side of the convolved image. This is often termed as circular convolution or cyclic convolution.

So, as long as we are dealing with convolution in the frequency domain, there is nothing we can do about the image edges. The least we can do is to eliminate the ghosts of the other side of the image. So, we add zero valued pixels to both the input image and the kernel in both dimensions so the image that will be convolved has a size equal to the sum of both images in each dimension. Of course, the effect of this zero-padding is that the sides of the output convolved image will become dark. To put it another way, the edges are going to drain the flux from nearby objects. But at least it is consistent across all the edges of the image and is predictable. In Convolve, you can see the padded images when inspecting the frequency domain convolution steps with the --viewfreqsteps option.

6.3.3 Spatial vs. Frequency domain

With the discussions above it might not be clear when to choose the spatial domain and when to choose the frequency domain. Here we will try to list the benefits of each.
The spatial domain,
- Can correct for the edge effects of convolution, see Section 6.3.1.2 [Edges in the spatial domain], page 307.
- Can operate on blank pixels.
- Can be faster than frequency domain when the kernel is small (in terms of the number of pixels on the sides).

The frequency domain,
- Will be much faster when the image and kernel are both large.

As a general rule of thumb, when working on an image of modeled profiles use the frequency domain and when working on an image of real (observed) objects use the spatial domain (corrected for the edges). The reason is that if you apply a frequency domain convolution to a real image, you are going to lose information on the edges and generally you don’t want large kernels. But when you have made the profiles in the image yourself, you can just make a larger input image and crop the central parts to completely remove the edge effect, see Section 8.1.2 [If convolving afterwards], page 446. Also due to oversampling, both the kernels and the images can become very large and the speed boost of frequency domain convolution will significantly improve the processing time, see Section 8.1.1.6 [Oversampling], page 445.

6.3.4 Convolution kernel

All the programs that need convolution will need to be given a convolution kernel file and extension. In most cases (other than Convolve, see Section 6.3 [Convolve], page 306) the kernel file name is optional. However, the extension is necessary and must be specified either on the command-line or at least one of the configuration files (see Section 4.2 [Configuration files], page 169). Within Gnuastro, there are two ways to create a kernel image:

- MakeProfiles: You can use MakeProfiles to create a parametric (based on a radial function) kernel, see Section 8.1 [MakeProfiles], page 440. By default MakeProfiles will make the Gaussian and Moffat profiles in a separate file so you can feed it into any of the programs.
- ConvertType: You can write your own desired kernel into a text file table and convert it to a FITS file with ConvertType, see Section 5.2 [ConvertType], page 211. Just be careful that the kernel has to have an odd number of pixels along its two axes, see Section 6.3.1.1 [Convolution process], page 307. All the programs that do convolution will normalize the kernel internally, so if you choose this option, you don’t have to worry about normalizing the kernel. Only within Convolve, there is an option to disable normalization, see Section 6.3.5 [Invoking Convolve], page 325.

The two options to specify a kernel file name and its extension are shown below. These are common between all the programs that will do convolution.

-k FITS
--kernel=FITS

The convolution kernel file name. The BITPIX (data type) value of this file can be any standard type and it does not necessarily have to be normalized. Several operations will be done on the kernel image prior to the program’s processing:
- It will be converted to floating point type.
• All blank pixels (see Section 6.1.3 [Blank pixels], page 262) will be set to zero.
• It will be normalized so the sum of its pixels equal unity.
• It will be flipped so the convolved image has the same orientation. This is only relevant if the kernel is not circular. See Section 6.3.1.1 [Convolution process], page 307.

-U STR
--khdu=STR

The convolution kernel HDU. Although the kernel file name is optional, before running any of the programs, they need to have a value for --khdu even if the default kernel is to be used. So be sure to keep its value in at least one of the configuration files (see Section 4.2 [Configuration files], page 169). By default, the system configuration file has a value.

6.3.5 Invoking Convolve

Convolve an input dataset (2D image or 1D spectrum for example) with a known kernel, or make the kernel necessary to match two PSFs. The general template for Convolve is:

$ astconvolve [OPTION...] ASTRdata

One line examples:

## Convolve mockimg.fits with psf.fits:
$ astconvolve --kernel=psf.fits mockimg.fits

## Convolve in the spatial domain:
$ astconvolve observedimg.fits --kernel=psf.fits --domain=spatial

## Find the kernel to match sharper and blurry PSF images (they both have to have the same pixel size).
$ astconvolve --kernel=sharperimage.fits --makekernel=10 \ blurryimage.fits

## Convolve a Spectrum (column 14 in the FITS table below) with a custom kernel (the kernel will be normalized internally, so only the ratios are important). Sed is used to replace the spaces with new line characters so Convolve sees them as values in one column.
$ echo "1 3 10 3 1" | sed 's/ /\n/g' | astconvolve spectra.fits -c14

The only argument accepted by Convolve is an input image file. Some of the options are the same between Convolve and some other Gnuastro programs. Therefore, to avoid repetition, they will not be repeated here. For the full list of options shared by all Gnuastro programs, please see Section 4.1.2 [Common options], page 156. In particular, in the spatial domain, on a multi-dimensional datasets, convolve uses Gnuastro’s tessellation to speed up the run, see Section 4.8 [Tessellation], page 189. Common options related to tessellation are described in Section 4.1.2.2 [Processing options], page 159.

1-dimensional datasets (for example spectra) are only read as columns within a table (see Section 4.7 [Tables], page 183, for more on how Gnuastro programs read tables). Note
that currently 1D convolution is only implemented in the spatial domain and thus kernel-matching is also not supported.

Here we will only explain the options particular to Convolve. Run Convolve with --help in order to see the full list of options Convolve accepts, irrespective of where they are explained in this book.

--kernelcolumn
Column containing the 1D kernel. When the input dataset is a 1-dimensional column, and the host table has more than one column, use this option to specify which column should be used.

--nokernelflip
Do not flip the kernel after reading it the spatial domain convolution. This can be useful if the flipping has already been applied to the kernel.

--nokernelnormx
Do not normalize the kernel after reading it, such that the sum of its pixels is unity.

-d STR
--domain=STR
The domain to use for the convolution. The acceptable values are ‘spatial’ and ‘frequency’, corresponding to the respective domain.

For large images, the frequency domain process will be more efficient than convolving in the spatial domain. However, the edges of the image will loose some flux (see Section 6.3.1.2 [Edges in the spatial domain], page 307) and the image must not contain any blank pixels, see Section 6.3.3 [Spatial vs. Frequency domain], page 323.

--checkfreqsteps
With this option a file with the initial name of the output file will be created that is suffixed with _freqsteps.fits, all the steps done to arrive at the final convolved image are saved as extensions in this file. The extensions in order are:

1. The padded input image. In frequency domain convolution the two images (input and convolved) have to be the same size and both should be padded by zeros.
2. The padded kernel, similar to the above.
3. The Fourier spectrum of the forward Fourier transform of the input image. Note that the Fourier transform is a complex operation (and not view able in one image!) So we either have to show the ‘Fourier spectrum’ or the ‘Phase angle’. For the complex number $a + ib$, the Fourier spectrum is defined as $\sqrt{a^2 + b^2}$ while the phase angle is defined as $\arctan(b/a)$.
4. The Fourier spectrum of the forward Fourier transform of the kernel image.
5. The Fourier spectrum of the multiplied (through complex arithmetic) transformed images.
6. The inverse Fourier transform of the multiplied image. If you open it, you will see that the convolved image is now in the center, not on one side of
the image as it started with (in the padded image of the first extension). If you are working on a mock image which originally had pixels of precisely 0.0, you will notice that in those parts that your convolved profile(s) did not convert, the values are now $\sim 10^{-18}$, this is due to floating-point round off errors. Therefore in the final step (when cropping the central parts of the image), we also remove any pixel with a value less than $10^{-17}$.

--noedgecorrection

Do not correct the edge effect in spatial domain convolution. For a full discussion, please see Section 6.3.1.2 [Edges in the spatial domain], page 307.

-m INT
--makekernel=INT

If this option is called, Convolve will do PSF-matching: the output will be the kernel that you should convolve with the sharper image to obtain the blurry one (see Section 6.3.2.6 [Convolution theorem], page 316). The two images must have the same size (number of pixels). This option is not yet supported in 1-dimensional datasets. In effect, it is only necessary to give the two PSFs of your two datasets, find the matching kernel based on that, then apply that kernel to the higher-resolution (sharper image).

The image given to the --kernel option is assumed to be the sharper (less blurry) image and the input image (with no option) is assumed to be the more blurry image. The value given to this option will be used as the maximum radius of the kernel. Any pixel in the final kernel that is larger than this distance from the center will be set to zero.

Noise has large frequencies which can make the result less reliable for the higher frequencies of the final result. So all the frequencies which have a spectrum smaller than the value given to the minsharpspec option in the sharper input image are set to zero and not divided. This will cause the wings of the final kernel to be flatter than they would ideally be which will make the convolved image result unreliable if it is too high.

Some notes to on how to prepare your two input PSFs. Note that these (and several other issues that relate to an accurate estimation of the PSF) are practically described in the following tutorial: Section 2.4 [Building the extended PSF], page 97.

- Choose a bright (unsaturated) star and use a region box (with Crop for example, see Section 6.1 [Crop], page 259) that is sufficiently above the noise.
- Mask all background sources that may be nearby (you can use Segment’s clumps, see Section 7.3 [Segment], page 381).
- Use Warp (see Section 6.4 [Warp], page 328) to warp the pixel grid so the star’s center is exactly on the center of the central pixel in the cropped image. This will certainly slightly degrade the result, however, it is necessary. If there are multiple good stars, you can shift all of them, then normalize them (so the sum of each star’s pixels is one) and then take their average to decrease this effect.
• The shifting might move the center of the star by one pixel in any direction, so crop the central pixel of the warped image to have a clean image for the de-convolution.

\[ \text{--minsharpspec} \] (\texttt{=FLT}) The minimum frequency spectrum (or coefficient, or pixel value in the frequency domain image) to use in deconvolution, see the explanations under the \texttt{--makekernel} option for more information.

\section*{6.4 Warp}

Image warping is the process of mapping the pixels of one image onto a new pixel grid. This process is sometimes known as transformation, however following the discussion of Heckbert 1989\textsuperscript{23} we will not be using that term because it can be confused with only pixel value or flux transformations. Here we specifically mean the pixel grid transformation which is better conveyed with ‘warp’.

Image wrapping is a very important step in astronomy, both in observational data analysis and in simulating modeled images. In modeling, warping an image is necessary when we want to apply grid transformations to the initial models, for example in simulating gravitational lensing (Radial warpings are not yet included in Warp). Observational reasons for warping an image are listed below:

• **Noise:** Most scientifically interesting targets are inherently faint (have a very low Signal to noise ratio). Therefore one short exposure is not enough to detect such objects that are drowned deeply in the noise. We need multiple exposures so we can add them together and increase the objects’ signal to noise ratio. Keeping the telescope fixed on one field of the sky is practically impossible. Therefore very deep observations have to put into the same grid before adding them.

• **Resolution:** If we have multiple images of one patch of the sky (hopefully at multiple orientations) we can warp them to the same grid. The multiple orientations will allow us to ‘guess’ the values of pixels on an output pixel grid that has smaller pixel sizes and thus increase the resolution of the output. This process of merging multiple observations is known as Mosaicing.

• **Cosmic rays:** Cosmic rays can randomly fall on any part of an image. If they collide vertically with the camera, they are going to create a very sharp and bright spot that in most cases can be separated easily\textsuperscript{24}. However, depending on the depth of the camera pixels, and the angle that a cosmic rays collides with it, it can cover a line-like larger area on the CCD which makes the detection using their sharp edges very hard and error prone. One of the best methods to remove cosmic rays is to compare multiple images of the same field. To do that, we need all the images to be on the same pixel grid.

• **Optical distortion:** (Not yet included in Warp) In wide field images, the optical distortion that occurs on the outer parts of the focal plane will make accurate comparison of


\textsuperscript{24} All astronomical targets are blurred with the PSF, see Section 8.1.1.2 [Point spread function], page 442, however a cosmic ray is not and so it is very sharp (it suddenly stops at one pixel).
the objects at various locations impossible. It is therefore necessary to warp the image and correct for those distortions prior to the analysis.

- **Detector not on focal plane:** In some cases (like the Hubble Space Telescope ACS and WFC3 cameras), the CCD might be tilted compared to the focal plane, therefore the recorded CCD pixels have to be projected onto the focal plane before further analysis.

### 6.4.1 Warping basics

Let’s take \([u \ v]\) as the coordinates of a point in the input image and \([x \ y]\) as the coordinates of that same point in the output image\(^{25}\). The simplest form of coordinate transformation (or warping) is the scaling of the coordinates, let’s assume we want to scale the first axis by \(M\) and the second by \(N\), the output coordinates of that point can be calculated by

\[
\begin{bmatrix}
x \\
y
\end{bmatrix} = \begin{bmatrix} Mu \\ Nv \end{bmatrix} = \begin{bmatrix} M & 0 \\ 0 & N \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix}
\]

Note that these are matrix multiplications. We thus see that we can represent any such grid warping as a matrix. Another thing we can do with this \(2 \times 2\) matrix is to rotate the output coordinate around the common center of both coordinates. If the output is rotated anticlockwise by \(\theta\) degrees from the positive (to the right) horizontal axis, then the warping matrix should become:

\[
\begin{bmatrix}
x \\
y
\end{bmatrix} = \begin{bmatrix} u\cos\theta - v\sin\theta \\ u\sin\theta + v\cos\theta \end{bmatrix} = \begin{bmatrix} \cos\theta & -\sin\theta \\ \sin\theta & \cos\theta \end{bmatrix} \begin{bmatrix} u \\ v \end{bmatrix}
\]

We can also flip the coordinates around the first axis, the second axis and the coordinate center with the following three matrices respectively:

\[
\begin{bmatrix} 1 & 0 \\ 0 & -1 \end{bmatrix}, \begin{bmatrix} -1 & 0 \\ 0 & 1 \end{bmatrix}, \begin{bmatrix} -1 & 0 \\ 0 & -1 \end{bmatrix}
\]

The final thing we can do with this definition of a \(2 \times 2\) warping matrix is shear. If we want the output to be sheared along the first axis with \(A\) and along the second with \(B\), then we can use the matrix:

\[
\begin{bmatrix} 1 & A \\ B & 1 \end{bmatrix}
\]

To have one matrix representing any combination of these steps, you use matrix multiplication, see Section 6.4.2 [Merging multiple warpings], page 331. So any combinations of these transformations can be displayed with one \(2 \times 2\) matrix:

\[
\begin{bmatrix} a & b \\ c & d \end{bmatrix}
\]

\(^{25}\) These can be any real number, we are not necessarily talking about integer pixels here.
The transformations above can cover a lot of the needs of most coordinate transformations. However, they are limited to mapping the point \([0, 0]\) to \([0, 0]\). Therefore, they are useless if you want one coordinate to be shifted compared to the other one. They are also space-invariant, meaning that all the coordinates in the image will receive the same transformation. In other words, all the pixels in the output image will have the same area if placed over the input image. So transformations which require varying output pixel sizes like projections cannot be applied through this \(2 \times 2\) matrix either (for example for the tilted ACS and WFC3 camera detectors on board the Hubble space telescope).

To add these further capabilities, namely translation and projection, we use the homogeneous coordinates. They were defined about 200 years ago by August Ferdinand Möbius (1790 – 1868). For simplicity, we will only discuss points on a 2D plane and avoid the complexities of higher dimensions. We cannot provide a deep mathematical introduction here, interested readers can get a more detailed explanation from Wikipedia\(^{26}\) and the references therein.

By adding an extra coordinate to a point we can add the flexibility we need. The point \([x, y]\) can be represented as \([xZ, yZ, Z]\) in homogeneous coordinates. Therefore, multiplying all the coordinates of a point in the homogeneous coordinates with a constant will give the same point. Put another way, the point \([x, y, Z]\) corresponds to the point \([x/Z, y/Z]\) on the constant \(Z\) plane. Setting \(Z = 1\), we get the input image plane, so \([u, v, 1]\) corresponds to \([u, v]\). With this definition, the transformations above can be generally written as:

\[
\begin{bmatrix}
x \\
y \\
1
\end{bmatrix}
= 
\begin{bmatrix}
a & b & 0 \\
c & d & 0 \\
0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
u \\
v
\end{bmatrix}
\]

We thus acquired 4 extra degrees of freedom. By giving non-zero values to the zero valued elements of the last column we can have translation (try the matrix multiplication!). In general, any coordinate transformation that is represented by the matrix below is known as an affine transformation\(^{27}\):

\[
\begin{bmatrix}
a & b & c \\
d & e & f \\
0 & 0 & 1
\end{bmatrix}
\]

We can now consider translation, but the affine transform is still spatially invariant. Giving non-zero values to the other two elements in the matrix above gives us the projective transformation or Homography\(^{28}\) which is the most general type of transformation with the \(3 \times 3\) matrix:

\[
\begin{bmatrix}
x' \\
y' \\
w
\end{bmatrix}
= 
\begin{bmatrix}
a & b & c \\
d & e & f \\
g & h & 1
\end{bmatrix}
\begin{bmatrix}
u \\
v
\end{bmatrix}
\]

\(^{26}\) http://en.wikipedia.org/wiki/Homogeneous_coordinates

\(^{27}\) http://en.wikipedia.org/wiki/Affine_transformation

\(^{28}\) http://en.wikipedia.org/wiki/Homography
So the output coordinates can be calculated from:

\[ x = \frac{x'}{w} = \frac{au + bv + c}{gu + hv + 1} \quad y = \frac{y'}{w} = \frac{du + ev + f}{gu + hv + 1} \]

Thus with Homography we can change the sizes of the output pixels on the input plane, giving a ‘perspective’-like visual impression. This can be quantitatively seen in the two equations above. When \( g = h = 0 \), the denominator is independent of \( u \) or \( v \) and thus we have spatial invariance. Homography preserves lines at all orientations. A very useful fact about Homography is that its inverse is also a Homography. These two properties play a very important role in the implementation of this transformation. A short but instructive and illustrated review of affine, projective and also bi-linear mappings is provided in Heckbert 1989\(^{29}\).

### 6.4.2 Merging multiple warpings

In Section 6.4.1 [Warping basics], page 329, we saw how a basic warp/transformation can be represented with a matrix. To make more complex warpings (for example to define a translation, rotation and scale as one warp) the individual matrices have to be multiplied through matrix multiplication. However matrix multiplication is not commutative, so the order of the set of matrices you use for the multiplication is going to be very important.

The first warping should be placed as the left-most matrix. The second warping to the right of that and so on. The second transformation is going to occur on the warped coordinates of the first. As an example for merging a few transforms into one matrix, the multiplication below represents the rotation of an image about a point \([U \ V]\) anticlockwise from the horizontal axis by an angle of \(\theta\). To do this, first we take the origin to \([U \ V]\) through translation. Then we rotate the image, then we translate it back to where it was initially. These three operations can be merged in one operation by calculating the matrix multiplication below:

\[
\begin{bmatrix}
1 & 0 & U \\
0 & 1 & V \\
0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
\cos\theta & -\sin\theta & 0 \\
\sin\theta & \cos\theta & 0 \\
0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
1 & 0 & -U \\
0 & 1 & -V \\
0 & 0 & 1
\end{bmatrix}
\]

### 6.4.3 Resampling

A digital image is composed of discrete ‘picture elements’ or ‘pixels’. When a real image is created from a camera or detector, each pixel’s area is used to store the number of photo-electrons that were created when incident photons collided with that pixel’s surface area. This process is called the ‘sampling’ of a continuous or analog data into digital data. When we change the pixel grid of an image or warp it as we defined in Section 6.4.1 [Warping basics], page 329, we have to ‘guess’ the flux value of each pixel on the new grid based on the old grid, or re-sample it. Because of the ‘guessing’, any form of warping on the data

\(^{29}\) Paul S. Heckbert. 1989. *Fundamentals of Texture mapping and Image Warping*, Master’s thesis at University of California, Berkeley. Note that since points are defined as row vectors there, the matrix is the transpose of the one discussed here.
is going to degrade the image and mix the original pixel values with each other. So if an
analysis can be done on an unwarped data image, it is best to leave the image untouched
and pursue the analysis. However as discussed in Section 6.4 [Warp], page 328, this is not
possible most of the times, so we have to accept the problem and re-sample the image.

In most applications of image processing, it is sufficient to consider each pixel to be a
point and not an area. This assumption can significantly speed up the processing of an
image and also the simplicity of the code. It is a fine assumption when the signal to noise
ratio of the objects are very large. The question will then be one of interpolation because
you have multiple points distributed over the output image and you want to find the values
at the pixel centers. To increase the accuracy, you might also sample more than one point
from within a pixel giving you more points for a more accurate interpolation in the output
grid.

However, interpolation has several problems. The first one is that it will depend on the
type of function you want to assume for the interpolation. For example you can choose a
bi-linear or bi-cubic (the ‘bi’s are for the 2 dimensional nature of the data) interpolation
method. For the latter there are various ways to set the constants\textsuperscript{30}. Such functional
interpolation functions can fail seriously on the edges of an image. They will also need
normalization so that the flux of the objects before and after the warping are comparable.
The most basic problem with such techniques is that they are based on a point while
a detector pixel is an area. They add a level of subjectivity to the data (make more
assumptions through the functions than the data can handle). For most applications this is
fine, but in scientific applications where detection of the faintest possible galaxies or fainter
parts of bright galaxies is our aim, we cannot afford this loss. Because of these reasons
Warp will not use such interpolation techniques.

Warp will do interpolation based on “pixel mixing”\textsuperscript{31} or “area resampling”. This is
also what the Hubble Space Telescope pipeline calls “Drizzling”\textsuperscript{32}. This technique requires
no functions, it is thus non-parametric. It is also the closest we can get (make least as-
sumptions) to what actually happens on the detector pixels. The basic idea is that you
reverse-transform each output pixel to find which pixels of the input image it covers and
what fraction of the area of the input pixels are covered. To find the output pixel value,
you simply sum the value of each input pixel weighted by the overlapfraction (between 0
to 1) of the output pixel and that input pixel. Through this process, pixels are treated as
an area not as a point (which is how detectors create the image), also the brightness (see
Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394)
of an object
will be left completely unchanged.

If there are very high spatial-frequency signals in the image (for example fringes) which
vary on a scale smaller than your output image pixel size, pixel mixing can cause ailing\textsuperscript{33}. So if the input image has fringes, they have to be calculated and removed separately (which
would naturally be done in any astronomical application). Because of the PSF no astrono-
mical target has a sharpchange in the signal so this issue is less important for astronomical
applications, see Section 8.1.1.2 [Point spread function], page 442.

\textsuperscript{30} see \url{http://entropymine.com/imageworsener/bicubic/} for a nice introduction.
\textsuperscript{31} For a graphic demonstration see \url{http://entropymine.com/imageworsener/pixelmixing/}.
\textsuperscript{32} \url{http://en.wikipedia.org/wiki/Drizzle_(image_processing)}
\textsuperscript{33} \url{http://en.wikipedia.org/wiki/Aliasing}
6.4.4 Invoking Warp

Warp an input dataset into a new grid. Any homographic warp (for example scaling, rotation, translation, projection) is acceptable, see Section 6.4.1 [Warping basics], page 329, for the definitions. The general template for invoking Warp is:

\$ astwarp [OPTIONS...] InputImage

One line examples:

## Rotate and then scale input image:
\$ astwarp --rotate=37.92 --scale=0.8 image.fits

## Scale, then translate the input image:
\$ astwarp --scale 8/3 --translate 2.1 image.fits

## Align raw image with celestial coordinates:
\$ astwarp --align rawimage.fits --output=aligned.fits

## Directly input a custom warping matrix (using fraction):
\$ astwarp --matrix=1/5,0,4/10,0,1/5,4/10,0,0,1 image.fits

## Directly input a custom warping matrix, with final numbers:
\$ astwarp --matrix="0.7071,-0.7071, 0.7071,0.7071" image.fits

If any processing is to be done, Warp can accept one file as input. As in all Gnuastro programs, when an output is not explicitly set with the `--output` option, the output file-name will be set automatically based on the operation, see Section 4.9 [Automatic output], page 191. For the full list of general options to all Gnuastro programs (including Warp), please see Section 4.1.2 [Common options], page 156.

To be the most accurate, the input image will be read as a 64-bit double precision floating point dataset and all internal processing is done in this format (including the raw output type). You can use the common `--type` option to write the output in any type you want, see Section 4.5 [Numeric data types], page 178.

Warp must be specified as command-line options, either as (possibly multiple) modular warpings (for example `--rotate`, or `--scale`), or directly as a single raw matrix (with `--matrix`). If specified together, the latter (direct matrix) will take precedence and all the modular warpings will be ignored. Any number of modular warpings can be specified on the command-line and configuration files. If more than one modular warping is given, all will be merged to create one warping matrix. As described in Section 6.4.2 [Merging multiple warpings], page 331, matrix multiplication is not commutative, so the order of specifying the modular warpings on the command-line, and/or configuration files makes a difference (see Section 4.2.2 [Configuration file precedence], page 170). The full list of modular warpings and the other options particular to Warp are described below.

The values to the warping options (modular warpings as well as `--matrix`), are a sequence of at least one number. Each number in this sequence is separated from the next by a comma (,). Each number can also be written as a single fraction (with a forward-slash / between the numerator and denominator). Space and Tab characters are permitted between any two numbers, just don’t forget to quote the whole value. Otherwise, the value will not be fully passed onto the option. See the examples above as a demonstration.
Based on the FITS standard, integer values are assigned to the center of a pixel and the coordinate \([1.0, 1.0]\) is the center of the first pixel (bottom left of the image when viewed in SAO DS9). So the coordinate center \([0.0, 0.0]\) is half a pixel away (in each axis) from the bottom left vertex of the first pixel. The resampling that is done in Warp (see Section 6.4.3 [Resampling], page 331) is done on the coordinate axes and thus directly depends on the coordinate center. In some situations this is fine, for example when rotating/aligning a real image, all the edge pixels will be similarly affected. But in other situations (for example when scaling an over-sampled mock image to its intended resolution, this is not desired: you want the center of the coordinates to be on the corner of the pixel. In such cases, you can use the \(--\text{centeroncorner}\) option which will shift the center by 0.5 before the main warp, then shift it back by \(-0.5\) after the main warp, see below.

\[-a\]

\(--\text{align}\) Align the image and celestial (WCS) axes given in the input. After it, the vertical image direction (when viewed in SAO DS9) corresponds to the declination and the horizontal axis is the inverse of the Right Ascension (RA). The inverse of the RA is chosen so the image can correspond to what you would actually see on the sky and is common in most survey images.

Align is internally treated just like a rotation \(--\text{rotation}\), but uses the input image’s WCS to find the rotation angle. Thus, if you have rotated the image before calling \(--\text{align}\), you might get unexpected results (because the rotation is defined on the original WCS).

\[-r\ \text{FLT}\]

\(--\text{rotate}=\text{FLT}\)

Rotate the input image by the given angle in degrees: \(\theta\) in Section 6.4.1 [Warping basics], page 329. Note that commonly, the WCS structure of the image is set such that the RA is the inverse of the image horizontal axis which increases towards the right in the FITS standard and as viewed by SAO DS9. So the default center for rotation is on the right of the image. If you want to rotate about other points, you have to translate the warping center first (with \(--\text{translate}\) then apply your rotation and then return the center back to the original position (with another call to \(--\text{translate}\), see Section 6.4.2 [Merging multiple warpings], page 331.

\[-s\ \text{FLT},\text{FLT}\]

\(--\text{scale}=\text{FLT},\text{FLT}\)

Scale the input image by the given factor(s): \(M\) and \(N\) in Section 6.4.1 [Warping basics], page 329. If only one value is given, then both image axes will be scaled with the given value. When two values are given (separated by a comma), the first will be used to scale the first axis and the second will be used for the second axis. If you only need to scale one axis, use 1 for the axis you don’t need to scale. The value(s) can also be written (on the command-line or in configuration files) as a fraction.

\[-f\ \text{FLT},\text{FLT}\]

\(--\text{flip}=\text{FLT},\text{FLT}\)

Flip the input image around the given axis(s). If only one value is given, then both image axes are flipped. When two values are given (separated by
acomma), you can choose which axis to flip over.  The `--flip` only takes values 0 (for no flip), or 1 (for a flip). Hence, if you want to flip by the second axis only, use `--flip=0,1`.

`-e FLT[,FLT]`  
`--shear=FLT[,FLT]`  
Shear the input image by the given value(s): \(A\) and \(B\) in Section 6.4.1 [Warping basics], page 329. If only one value is given, then both image axes will be sheared with the given value. When two values are given (separated by a comma), the first will be used to shear the first axis and the second will be used for the second axis. If you only need to shear along one axis, use 0 for the axis that must be untouched. The value(s) can also be written (on the command-line or in configuration files) as a fraction.

`-t FLT[,FLT]`  
`--translate=FLT[,FLT]`  
Translate (move the center of coordinates) the input image by the given value(s): \(c\) and \(f\) in Section 6.4.1 [Warping basics], page 329. If only one value is given, then both image axes will be translated by the given value. When two values are given (separated by a comma), the first will be used to translate the first axis and the second will be used for the second axis. If you only need to translate along one axis, use 0 for the axis that must be untouched. The value(s) can also be written (on the command-line or in configuration files) as a fraction.

`-p FLT[,FLT]`  
`--project=FLT[,FLT]`  
Apply a projection to the input image by the given values(s): \(g\) and \(h\) in Section 6.4.1 [Warping basics], page 329. If only one value is given, then projection will apply to both axes with the given value. When two values are given (separated by a comma), the first will be used to project the first axis and the second will be used for the second axis. If you only need to project along one axis, use 0 for the axis that must be untouched. The value(s) can also be written (on the command-line or in configuration files) as a fraction.

`-m STR`  
`--matrix=STR`  
The warp/transformation matrix. All the elements in this matrix must be separated by commas(,) characters and as described above, you can also use fractions (a forward-slash between two numbers). The transformation matrix can be either a 2 by 2 (4 numbers), or a 3 by 3 (9 numbers) array. In the former case (if a 2 by 2 matrix is given), then it is put into a 3 by 3 matrix (see Section 6.4.1 [Warping basics], page 329).

The determinant of the matrix has to be non-zero and it must not contain any non-number values (for example infinities or NaNs). The elements of the matrix have to be written row by row. So for the general Homography matrix of Section 6.4.1 [Warping basics], page 329, it should be called with `--matrix=a,b,c,d,e,f,g,h,1`. The raw matrix takes precedence over all the modular warping options listed above, so if it is called with any number of modular warps, the latter are ignored.
-c
--centeroncorner
Put the center of coordinates on the corner of the first (bottom-left when viewed in SAO DS9) pixel. This option is applied after the final warping matrix has been finalized: either through modular warpings or the raw matrix. See the explanation above for coordinates in the FITS standard to better understand this option and when it should be used.

--hstartwcs=INT
Specify the first header keyword number (line) that should be used to read the WCS information, see the full explanation in Section 6.1.4 [Invoking Crop], page 263.

--hendwcs=INT
Specify the last header keyword number (line) that should be used to read the WCS information, see the full explanation in Section 6.1.4 [Invoking Crop], page 263.

-k
--keepwcs
Do not correct the WCS information of the input image and save it untouched to the output image. By default the WCS (World Coordinate System) information of the input image is going to be corrected in the output image so the objects in the image are at the same WCS coordinates. But in some cases it might be useful to keep it unchanged (for example to correct alignments).

-C FLT
--coveredfrac=FLT
Depending on the warp, the output pixels that cover pixels on the edge of the input image, or blank pixels in the input image, are not going to be fully covered by input data. With this option, you can specify the acceptable covered fraction of such pixels (any value between 0 and 1). If you only want output pixels that are fully covered by the input image area (and are not blank), then you can set --coveredfrac=1. Alternatively, a value of 0 will keep output pixels that are even infinitesimally covered by the input (so the sum of the pixels in the input and output images will be the same).
7 Data analysis

Astronomical datasets (images or tables) contain very valuable information, the tools in this section can help in analyzing, extracting, and quantifying that information. For example getting general or specific statistics of the dataset (with Section 7.1 [Statistics], page 337), detecting signal within a noisy dataset (with Section 7.2 [NoiseChisel], page 362), or creating a catalog from an input dataset (with Section 7.4 [MakeCatalog], page 392).

7.1 Statistics

The distribution of values in a dataset can provide valuable information about it. For example, in an image, if it is a positively skewed distribution, we can see that there is significant data in the image. If the distribution is roughly symmetric, we can tell that there is no significant data in the image. In a table, when we need to select a sample of objects, it is important to first get a general view of the whole sample.

On the other hand, you might need to know certain statistical parameters of the dataset. For example, if we have run a detection algorithm on an image, and we want to see how accurate it was, one method is to calculate the average of the undetected pixels and see how reasonable it is (if detection is done correctly, the average of undetected pixels should be approximately equal to the background value, see Section 7.1.4 [Sky value], page 344). In a table, you might have calculated the magnitudes of a certain class of objects and want to get some general characteristics of the distribution immediately on the command-line (very fast!), to possibly change some parameters. The Statistics program is designed for such situations.

7.1.1 Histogram and Cumulative Frequency Plot

Histograms and the cumulative frequency plots are both used to visually study the distribution of a dataset. A histogram shows the number of data points which lie within pre-defined intervals (bins). So on the horizontal axis we have the bin centers and on the vertical, the number of points that are in that bin. You can use it to get a general view of the distribution: which values have been repeated the most? how close/far are the most significant bins? Are there more values in the larger part of the range of the dataset, or in the lower part? Similarly, many very important properties about the dataset can be deduced from a visual inspection of the histogram. In the Statistics program, the histogram can be either output to a table to plot with your favorite plotting program\(^1\), or it can be shown with ASCII characters on the command-line, which is very crude, but good enough for a fast and on-the-go analysis, see the example in Section 7.1.5 [Invoking Statistics], page 350.

The width of the bins is only necessary parameter for a histogram. In the limiting case that the bin-widths tend to zero (while assuming the number of points in the dataset tend to infinity), then the histogram will tend to the probability density function (https://en.wikipedia.org/wiki/Probability_density_function) of the distribution. When the absolute number of points in each bin is not relevant to the study (only the shape of the histogram is important), you can normalize a histogram so like the probability density function, the sum of all its bins will be one.

\(^1\) We recommend PGFPLOTS (http://pgfplots.sourceforge.net/) which generates your plots directly within \texttt{Ti\TeX} (the same tool that generates your document).
In the cumulative frequency plot of a distribution, the horizontal axis is the sorted data values and the y axis is the index of each data in the sorted distribution. Unlike a histogram, a cumulative frequency plot does not involve intervals or bins. This makes it less prone to any sort of bias or error that a given bin-width would have on the analysis. When a larger number of the data points have roughly the same value, then the cumulative frequency plot will become steep in that vicinity. This occurs because on the horizontal axis, there is little change while on the vertical axis, the indexes constantly increase. Normalizing a cumulative frequency plot means to divide each index (y axis) by the total number of data points (or the last value).

Unlike the histogram which has a limited number of bins, ideally the cumulative frequency plot should have one point for every data element. Even in small datasets (for example a 200 $\times$ 200 image) this will result in an unreasonably large number of points to plot (40000)! As a result, for practical reasons, it is common to only store its value on a certain number of points (intervals) in the input range rather than the whole dataset, so you should determine the number of bins you want when asking for a cumulative frequency plot. In Gnuastro (and thus the Statistics program), the number reported for each bin is the total number of data points until the larger interval value for that bin. You can see an example histogram and cumulative frequency plot of a single dataset under the `--asciihat` and `--asciicfp` options of Section 7.1.5 [Invoking Statistics], page 350.

So as a summary, both the histogram and cumulative frequency plot in Statistics will work with bins. Within each bin/interval, the lower value is considered to be within then bin (it is inclusive), but its larger value is not (it is exclusive). Formally, an interval/bin between a and b is represented by $[a, b)$. When the over-all range of the dataset is specified (with the `--greaterequal`, `--lessthan`, or `--qrange` options), the acceptable values of the dataset are also defined with a similar inclusive-exclusive manner. But when the range is determined from the actual dataset (none of these options is called), the last element in the dataset is included in the last bin’s count.

### 7.1.2 2D Histograms

In Section 7.1.1 [Histogram and Cumulative Frequency Plot], page 337, the concept of histograms were introduced on a single dataset. But they are only useful for viewing the distribution of a single variable (column in a table). In many contexts, the distribution of two variables in relation to each other may be of interest. For example the color-magnitude diagrams in astronomy, where the horizontal axis is the luminosity or magnitude of an object, and the vertical axis is the color. Scatter plots are useful to see these relations between the objects of interest when the number of the objects is small.

As the density of points in the scatter plot increases, the points will fall over each other and just make a large connected region hide potentially interesting behaviors/correlations in the densest regions. This is where 2D histograms can become very useful. A 2D histogram is composed of 2D bins (boxes or pixels), just as a 1D histogram consists of 1D bins (lines). The number of points falling within each box/pixel will then be the value of that box. Added with a color-bar, you can now clearly see the distribution independent of the density of points (for example you can even visualize it in log-scale if you want).

Gnuastro’s Statistics program has the `--histogram2d` option for this task. It takes a single argument (either `table` or `image`) that specifies the format of the output 2D histogram. The two formats will be reviewed separately in the sub-sections below. But
let’s start with the generalities that are common to both (related to the input, not the output).

You can specify the two columns to be shown using the --column (or -c) option. So if you want to plot the color-magnitude diagram from a table with the MAG-R column on the horizontal and COLOR-G-R on the vertical column, you can use --column=MAG-r, COLOR-G-r. The number of bins along each dimension can be set with --nombres (for first input column) and --nombres2 (for second input column).

Without specifying any range, the full range of values will be used in each dimension. If you only want to focus on a certain interval of the values in the columns in any dimension you can use the --greaterequal and --lessthan options to limit the values along the first/horizontal dimension and --greaterequal2 and --lessthan2 options for the second/vertical dimension.

### 7.1.2.1 2D histogram as a table

When called with the --histogram=table option, Statistics will output a table file with three columns that have the information of every box as a column. If you asked for --nombres=N and --nombres2=M, all three columns will have $M \times N$ rows (one row for every box/pixel of the 2D histogram). The first and second columns are the position of the box along the first and second dimensions. The third column has the number of input points that fall within that box/pixel.

For example, you can make high-quality plots within your paper (using the same \LaTeX engine, thus blending very nicely with your text) using PGFPlots (https://ctan.org/pkg/pgfplots). Below you can see one such minimal example, using your favorite text editor, save it into a file, make the two small corrections in it, then run the commands shown at the top. This assumes that you have \LaTeX installed, if not the steps to install a minimally sufficient \LaTeX package on your system, see the respective section in Section 3.1.3 [Bootstrapping dependencies], page 124.

The two parts that need to be corrected are marked with '%% <--': the first one (XXXXXXXXX) should be replaced by the value to the --nombres option which is the number of bins along the first dimension. The second one (FILE.txt) should be replaced with the name of the file generated by Statistics.

%%% Replace 'XXXXXXXXXX' with your selected number of bins in the first
%%% dimension.
%%% Then run these commands to build the plot in a \LaTeX command.
%%% mkdir tikz
%%% pdflatex -shell-escape -halt-on-error plot.tex
\documentclass{article}

%%% Load PGFPlots and set it to build the figure separately in a 'tikz'
%%% directory (which has to exist before \LaTeX is run). This
%%% "externalization" is very useful to include the commands of multiple
%%% plots in the middle of your paper/report, but also have the plots
%%% separately to use in slides or other scenarios.
\usepackage{pgfplots}
\usetikzlibrary{external}
You can actually write a full paper here and include many figures! Feel free to change this text.

%% Define the colormap.
\pgfplotsset{
  /pgfplots/colormap={coldredux}{
    [1cm]
    rgb255(0cm)=(255,255,255)
    rgb255(2cm)=(0,192,255)
    rgb255(4cm)=(0,0,255)
    rgb255(6cm)=(0,0,0)
  }
}

%% Draw the plot.
\begin{tikzpicture}
\small
\begin{axis}[
  width=\textwidth,
  view={0}{90},
  colorbar horizontal,
  xlabel=X axis,
  ylabel=Y axis,
  ylabel shift=-0.1cm,
  colorbar style={at={(0,1.01)}, anchor=south west,}
  xticklabel pos=upper,]
  \addplot3[
    surf,
    shader=flat corner,
    mesh/ordering=rowwise,
    mesh/cols=XXXXXXXXX, %%% <-- Number of bins in 1st column.
  ] file {FILE.txt}; %%% <-- Name of aststatistics output.
\end{axis}
\end{tikzpicture}
\end{document}
7.1.2.2 2D histogram as an image

When called with the `--histogram=image` option, Statistics will output a FITS file with an image/array extension. If you asked for `--numbins=N` and `--numbins2=M` the image will have a size of $N \times M$ pixels (one pixel per 2D bin). Also, the FITS image will have a linear WCS that is scaled to the 2D bin size along each dimension. So when you hover your mouse over any part of the image with a FITS viewer (for example SAO DS9), besides the number of points in each pixel, you can directly also see “coordinates” of the pixels along the two axes. You can also use the optimized and fast FITS viewer features for many aspects of visually inspecting the distributions (which we won’t go into further).

For example let’s assume you want to derive the color-magnitude diagram (CMD) of the UVUDF survey (http://uvudf.ipac.caltech.edu). You can run the first command below to download the table with magnitudes of objects in many filters and run the second command to see general column metadata after it is downloaded.

```bash
$ asttable uvudf_rafelski_2015.fits.gz -i
```

Let’s assume you want to find the color to be between the F606W and F775W filters (roughly corresponding to the g and r filters in ground-based imaging). However, the original table doesn’t have color columns (there would be too many combinations!). Therefore you can use the Section 5.3.1 [Column arithmetic], page 230, feature of Gnuastro’s Table program for deriving a new table with the F775W magnitude in one column and the difference between the F606W and F775W on the other column. With the second command, you can see the actual values if you like.

```bash
$ asttable uvudf_rafelski_2015.fits.gz -c MAG_F775W \
  -c 'arith MAG_F606W MAG_F775W -' \
  --colmetadata=ARITH_1,F606W-F775W,"AB mag" -ocmd.fits
$ asttable cmd.fits
```

You can now construct your 2D histogram as a 100 $\times$ 100 pixel FITS image with this command (assuming you want F775W magnitudes between 22 and 30, colors between -1 and 3 and 100 bins in each dimension). Note that without the `--manualbinrange` option the range of each axis will be determined by the values within the columns (which may be larger or smaller than your desired large).

```bash
aststatistics cmd.fits -c MAG_F775W,F606W-F775W --histogram2d=image \
  --numbins=100 --greaterequal=22 --lessthan=30 \ 
  --numbins2=100 --greaterequal2=-1 --lessthan2=3 \ 
  --manualbinrange --output=cmd-2d-hist.fits
```

If you have SAO DS9, you can now open this FITS file as a normal FITS image, for example with the command below. Try hovering/zooming over the pixels: not only will you see the number of objects in the UVUDF catalog that fall in each bin, but you also see the F775W magnitude and color of that pixel also.

```bash
$ ds9 cmd-2d-hist.fits -cmap sls -zoom fit
```

With the first command below, you can activate the grid feature of DS9 to actually see the coordinate grid, as well as values on each line. With the second command, DS9 will even read the labels of the axises and use them to generate an almost publication-ready plot.

```bash
$ ds9 cmd-2d-hist.fits -cmap sls -zoom fit -grid yes
$ ds9 cmd-2d-hist.fits -cmap sls -zoom to fit -grid yes \
-grid type publication

If you are happy with the grid and coloring and etc, you can also use ds9 to save this as a JPEG image to directly use in your documents/slides with these extra DS9 options (DS9 will write the image to cmd-2d.jpeg and quit immediately afterwards):

$ ds9 cmd-2d-hist.fits -cmap sls -zoom 4 -grid yes \
-grid type publication -saveimage cmd-2d.jpeg -quit

This is good for a fast progress update. But for your paper or more official report, you want to show something with higher quality. For that, you can use the PGFPLOTS package in İ₄TeX to add axises in the same font as your text, sharp grids and many other elegant/powerful features (like over-plotting interesting points, lines and etc). But to load the 2D histogram into PGFPLOTS first you need to convert the FITS image into a more standard format, for example PDF. We’ll use Gnuastro’s Section 5.2 [ConvertType], page 211, for this, and use the sls-inverse color map (which will map the pixels with a value of zero to white):

$ astconvertt cmd-2d-hist.fits --colormap=sls-inverse \
   --borderwidth=0 -ocmd-2d-hist.pdf

Below you can see a minimally working example of how to add axis numbers, labels and a grid to the PDF generated above. Copy and paste the İ₄TeX code below into a plain-text file called cmd-report.tex Notice the xmin, xmax, ymin, ymax values and how they are the same as the range specified above.

\documentclass{article}
\usepackage{pgfplots}
\dimen\prevdepth=0
\begin{document}

You can write all you want here...

\begin{tikzpicture}
\begin{axis}[
    enlargelimits=false, grid, axis on top, width=\linewidth, height=\linewidth, xlabel={Magnitude (F775W)}, ylabel={Color (F606W-F775W)}]
\addplot graphics[xmin=22, xmax=30, ymin=-1, ymax=3] {cmd-2d-hist.pdf};
\end{axis}
\end{tikzpicture}
\end{document}

Run this command to build your PDF (assuming you have İ₄TeX and PGFPLOTS).

$ pdflatex cmd-report.tex
The improved quality, blending in with the text, vector-graphics resolution and other features make this plot pleasing to the eye, and let your readers focus on the main point of your scientific argument. PGFPlots can also build the PDF of the plot separately from the rest of the paper/report, see Section 7.1.2.1 [2D histogram as a table], page 339, for the necessary changes in the preamble.

7.1.3 Sigma clipping

Let’s assume that you have pure noise (centered on zero) with a clear Gaussian distribution (https://en.wikipedia.org/wiki/Normal_distribution), or see Section 8.2.1.1 [Photon counting noise], page 462. Now let’s assume you add very bright objects (signal) on the image which have a very sharp boundary. By a sharp boundary, we mean that there is a clear cutoff (from the noise) at the pixels the objects finish. In other words, at their boundaries, the objects do not fade away into the noise. In such a case, when you plot the histogram (see Section 7.1.1 [Histogram and Cumulative Frequency Plot], page 337) of the distribution, the pixels relating to those objects will be clearly separate from pixels that belong to parts of the image that did not have any signal (were just noise). In the cumulative frequency plot, after a steady rise (due to the noise), you would observe a long flat region were for a certain range of data (horizontal axis), there is no increase in the index (vertical axis).

Outliers like the example above can significantly bias the measurement of noise statistics. $\sigma$-clipping is defined as a way to avoid the effect of such outliers. In astronomical applications, cosmic rays (when they collide at a near normal incidence angle) are a very good example of such outliers. The tracks they leave behind in the image are perfectly immune to the blurring caused by the atmosphere and the aperture. They are also very energetic and so their borders are usually clearly separated from the surrounding noise. So $\sigma$-clipping is very useful in removing their effect on the data. See Figure 15 in Akhlaghi and Ichikawa, 2015 (https://arxiv.org/abs/1505.01664).

$\sigma$-clipping is defined as the very simple iteration below. In each iteration, the range of input data might decrease and so when the outliers have the conditions above, the outliers will be removed through this iteration. The exit criteria will be discussed below.

1. Calculate the standard deviation ($\sigma$) and median ($m$) of a distribution.
2. Remove all points that are smaller or larger than $m \pm \alpha \sigma$.
3. Go back to step 1, unless the selected exit criteria is reached.

The reason the median is used as a reference and not the mean is that the mean is too significantly affected by the presence of outliers, while the median is less affected, see Section 7.1.4.3 [Quantifying signal in a tile], page 347. As you can tell from this algorithm, besides the condition above (that the signal have clear high signal to noise boundaries) $\sigma$-clipping is only useful when the signal does not cover more than half of the full data set. If they do, then the median will lie over the outliers and $\sigma$-clipping might remove the pixels with no signal.

There are commonly two exit criteria to stop the $\sigma$-clipping iteration:

• When a certain number of iterations has taken place (second value to the --sclipparams option is larger than 1).
• When the new measured standard deviation is within a certain tolerance level of the old one (second value to the --slipparams option is less than 1). The tolerance level is defined by:

\[
\frac{\sigma_{\text{old}} - \sigma_{\text{new}}}{\sigma_{\text{new}}}
\]

The standard deviation is used because it is heavily influenced by the presence of outliers. Therefore the fact that it stops changing between two iterations is a sign that we have successfully removed outliers. Note that in each clipping, the dispersion in the distribution is either less or equal. So \( \sigma_{\text{old}} \geq \sigma_{\text{new}} \).

When working on astronomical images, objects like galaxies and stars are blurred by the atmosphere and the telescope aperture, therefore their signal sinks into the noise very gradually. Galaxies in particular do not appear to have a clear high signal to noise cutoff at all. Therefore \( \sigma \)-clipping will not be useful in removing their effect on the data.

To gauge if \( \sigma \)-clipping will be useful for your dataset, look at the histogram (see Section 7.1.1 [Histogram and Cumulative Frequency Plot], page 337). The ASCII histogram that is printed on the command-line with --asciihist is good enough in most cases.

7.1.4 Sky value

One of the most important aspects of a dataset is its reference value: the value of the dataset where there is no signal. Without knowing, and thus removing the effect of, this value it is impossible to compare the derived results of many high-level analyses over the dataset with other datasets (in the attempt to associate our results with the “real” world).

In astronomy, this reference value is known as the “Sky” value: the value that noise fluctuates around: where there is no signal from detectable objects or artifacts (for example galaxies, stars, planets or comets, star spikes or internal optical ghost). Depending on the dataset, the Sky value maybe a fixed value over the whole dataset, or it may vary based on location. For an example of the latter case, see Figure 11 in Akhlaghi and Ichikawa (2015) (https://arxiv.org/abs/1505.01664).

Because of the significance of the Sky value in astronomical data analysis, we have devoted this subsection to it for a thorough review. We start with a thorough discussion on its definition (Section 7.1.4.1 [Sky value definition], page 345). In the astronomical literature, researchers use a variety of methods to estimate the Sky value, so in Section 7.1.4.2 [Sky value misconceptions], page 346 we review those and discuss their biases. From the definition of the Sky value, the most accurate way to estimate the Sky value is to run a detection algorithm (for example Section 7.2 [NoiseChisel], page 362) over the dataset and use the undetected pixels. However, there is also a more crude method that maybe useful when good direct detection is not initially possible (for example due to too many cosmic rays in a shallow image). A more crude (but simpler method) that is usable in such situations is discussed in Section 7.1.4.3 [Quantifying signal in a tile], page 347.
7.1.4.1 Sky value definition

This analysis is taken from Akhlaghi and Ichikawa (2015) (https://arxiv.org/abs/1505.01664). Let’s assume that all instrument defects – bias, dark and flat – have been corrected and the brightness (see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394) of a detected object, \( O \), is desired. The sources of flux on pixel\(^2 \) \( i \) of the image can be written as follows:

- Contribution from the target object (\( O_i \)).
- Contribution from other detected objects (\( D_i \)).
- Undetected objects or the fainter undetected regions of bright objects (\( U_i \)).
- A cosmic ray (\( C_i \)).
- The background flux, which is defined to be the count if none of the others exists on that pixel (\( B_i \)).

The total flux in this pixel (\( T_i \)) can thus be written as:

\[
T_i = B_i + D_i + U_i + C_i + O_i.
\]

By definition, \( D_i \) is detected and it can be assumed that it is correctly estimated (deblended) and subtracted, we can thus set \( D_i = 0 \). There are also methods to detect and remove cosmic rays, for example the method described in van Dokkum (2001)\(^3 \), or by comparing multiple exposures. This allows us to set \( C_i = 0 \). Note that in practice, \( D_i \) and \( U_i \) are correlated, because they both directly depend on the detection algorithm and its input parameters. Also note that no detection or cosmic ray removal algorithm is perfect. With these limitations in mind, the observed Sky value for this pixel (\( S_i \)) can be defined as

\[
S_i = B_i + U_i.
\]

Therefore, as the detection process (algorithm and input parameters) becomes more accurate, or \( U_i \to 0 \), the Sky value will tend to the background value or \( S_i \to B_i \). Hence, we see that while \( B_i \) is an inherent property of the data (pixel in an image), \( S_i \) depends on the detection process. Over a group of pixels, for example in an image or part of an image, this equation translates to the average of undetected pixels (\( \text{Sky} = \sum S_i \)). With this definition of Sky, the object flux in the data can be calculated, per pixel, with

\[
T_i = S_i + O_i \quad \rightarrow \quad O_i = T_i - S_i.
\]

In the fainter outskirts of an object, a very small fraction of the photo-electrons in a pixel actually belongs to objects, the rest is caused by random factors (noise), see Figure 1b in Akhlaghi and Ichikawa (2015) (https://arxiv.org/abs/1505.01664). Therefore even

\(^2\) For this analysis the dimension of the data (image) is irrelevant. So if the data is an image (2D) with width of \( w \) pixels, then a pixel located on column \( x \) and row \( y \) (where all counting starts from zero and \( (0, 0) \) is located on the bottom left corner of the image), would have an index: \( i = x + y \times w \).

a small over estimation of the Sky value will result in the loss of a very large portion of most galaxies. Besides the lost area/brightness, this will also cause an over-estimation of the Sky value and thus even more under-estimation of the object’s brightness. It is thus very important to detect the diffuse flux of a target, even if they are not your primary target.

In summary, the more accurately the Sky is measured, the more accurately the brightness (sum of pixel values) of the target object can be measured (photometry). Any under/over-estimation in the Sky will directly translate to an over/under-estimation of the measured object’s brightness.

The Sky value is only correctly found when all the detected objects \((D_i \text{ and } C_i)\) have been removed from the data.

### 7.1.4.2 Sky value misconceptions

As defined in Section 7.1.4 [Sky value], page 344, the sky value is only accurately defined when the detection algorithm is not significantly reliant on the sky value. In particular its detection threshold. However, most signal-based detection tools\(^4\) use the sky value as a reference to define the detection threshold. These older techniques therefore had to rely on approximations based on other assumptions about the data. A review of those other techniques can be seen in Appendix A of Akhlaghi and Ichikawa (2015) ([https://arxiv.org/abs/1505.01664](https://arxiv.org/abs/1505.01664)).

These methods were extensively used in astronomical data analysis for several decades, therefore they have given rise to a lot of misconceptions, ambiguities and disagreements about the sky value and how to measure it. As a summary, the major methods used until now were an approximation of the mode of the image pixel distribution and \(\sigma\)-clipping.

- To find the mode of a distribution those methods would either have to assume (or find) a certain probability density function (PDF) or use the histogram. But astronomical datasets can have any distribution, making it almost impossible to define a generic function. Also, histogram-based results are very inaccurate (there is a large dispersion) and it depends on the histogram bin-widths. Generally, the mode of a distribution also shifts as signal is added. Therefore, even if it is accurately measured, the mode is a biased measure for the Sky value.

- Another approach was to iteratively clip the brightest pixels in the image (which is known as \(\sigma\)-clipping). See Section 7.1.3 [Sigma clipping], page 343, for a complete explanation. \(\sigma\)-clipping is useful when there are clear outliers (an object with a sharp edge in an image for example). However, real astronomical objects have diffuse and faint wings that penetrate deeply into the noise, see Figure 1 in Akhlaghi and Ichikawa (2015) ([https://arxiv.org/abs/1505.01664](https://arxiv.org/abs/1505.01664)).

As discussed in Section 7.1.4 [Sky value], page 344, the sky value can only be correctly defined as the average of undetected pixels. Therefore all such approaches that try to approximate the sky value prior to detection are ultimately poor approximations.

---

\(^4\) According to Akhlaghi and Ichikawa (2015), signal-based detection is a detection process that relies heavily on assumptions about the to-be-detected objects. This method was the most heavily used technique prior to the introduction of NoiseChisel in that paper.
7.1.4.3 Quantifying signal in a tile

In order to define detection thresholds on the image, or calibrate it for measurements (subtract the signal of the background sky and define errors), we need some basic measurements. For example the quantile threshold in NoiseChisel (\texttt{--qthresh} option), or the mean of the undetected regions (Sky) and the Sky standard deviation (Sky STD) which are the output of NoiseChisel and Statistics. But astronomical images will contain a lot of stars and galaxies that will bias those measurements if not properly accounted for. Quantifying where signal is present is thus a very important step in the usage of a dataset: for example if the Sky level is over-estimated, your target object’s brightness will be under-estimated.

Let’s start by clarifying some definitions: \textit{Signal} is defined as the non-random source of flux in each pixel (you can think of this as the mean in a Gaussian or Poisson distribution). In astronomical images, signal is mostly photons coming of a star or galaxy, and counted in each pixel. \textit{Noise} is defined as the random source of flux in each pixel (or the standard deviation of a Gaussian or Poisson distribution). Noise is mainly due to counting errors in the detector electronics upon data collection. \textit{Data} is defined as the combination of signal and noise (so a noisy image of a galaxy is one dataset).

When a dataset doesn’t have any signal (for example you take an image with a closed shutter, producing an image that only contains noise), the mean, median and mode of the distribution are equal within statistical errors. Signal from emitting objects, like astronomical targets, always has a positive value and will never become negative, see Figure 1 in Akhlaghi and Ichikawa (2015) \texttt{(https://arxiv.org/abs/1505.01664)}. Therefore, when signal is added to the data (you take an image with an open shutter pointing to a galaxy for example), the mean, median and mode of the dataset shift to the positive, creating a positively skewed distribution. The shift of the mean is the largest. The median shifts less, since it is defined after ordering all the elements/pixels (the median is the value at a quantile of 0.5), thus it is not affected by outliers. Finally, the mode’s shift to the positive is the least.

Inverting the argument above gives us a robust method to quantify the significance of signal in a dataset: when the mean and median of a distribution are approximately equal we can argue that there is no significant signal. In other words: when the quantile of the mean ($q_{\text{mean}}$) is around 0.5. This definition of skewness through the quantile of the mean is further introduced with a real image the tutorials, see Section 2.3.3 \texttt{[Skewness caused by signal and its measurement]}, page 83.

However, in an astronomical image, some of the pixels will contain more signal than the rest, so we can’t simply check $q_{\text{mean}}$ on the whole dataset. For example if we only look at the patch of pixels that are placed under the central parts of the brightest stars in the field of view, $q_{\text{mean}}$ will be very high. The signal in other parts of the image will be weaker, and in some parts it will be much smaller than the noise (for example 1/100-th of the noise level). When the signal-to-noise ratio is very small, we can generally assume no signal (because its effectively impossible to measure it) and $q_{\text{mean}}$ will be approximately 0.5.

To address this problem, we break the image into a grid of tiles\footnote{The options to customize the tessellation are discussed in Section 4.1.2.2 \texttt{[Processing options]}, page 159.} \texttt{(see Section 4.8 \texttt{[Tessellation]}, page 189)}. For example a tile can be a square box of size 30 $\times$ 30 pixels. By measuring $q_{\text{mean}}$ on each tile, we can find which tiles that contain significant signal and ignore them. Technically, if a tile’s $|q_{\text{mean}} - 0.5|$ is larger than the value given to the
--meanmedqdiff option, that tile will be ignored for the next steps. You can read this option as “mean-median-quantile-difference”.

The raw dataset’s pixel distribution (in each tile) is noisy, to decrease the noise/error in estimating $q_{\text{mean}}$, we convolve the image before tessellation (see Section 6.3.1.1 [Convolution process], page 307. Convolution decreases the range of the dataset and enhances its skewness, See Section 3.1.1 and Figure 4 in Akhlaghi and Ichikawa (2015) (https://arxiv.org/abs/1505.01664). This enhanced skewness can be interpreted as an increase in the Signal to noise ratio of the objects buried in the noise. Therefore, to obtain an even better measure of the presence of signal in a tile, the mean and median discussed above are measured on the convolved image.

There is one final hurdle: raw astronomical datasets are commonly peppered with Cosmic rays. Images of Cosmic rays aren’t smoothed by the atmosphere or telescope aperture, so they have sharp boundaries. Also, since they don’t occupy too many pixels, they don’t affect the mode and median calculation. But their very high values can greatly bias the calculation of the mean (recall how the mean shifts the fastest in the presence of outliers), for example see Figure 15 in Akhlaghi and Ichikawa (2015) (https://arxiv.org/abs/1505.01664). The effect of outliers like cosmic rays on the mean and standard deviation can be removed through $\sigma$-clipping, see Section 7.1.3 [Sigma clipping], page 343, for a complete explanation.

Therefore, after asserting that the mean and median are approximately equal in a tile (see Section 4.8 [Tessellation], page 189), the Sky and its STD are measured on each tile after $\sigma$-clipping with the --sigmaclip option (see Section 7.1.3 [Sigma clipping], page 343). In the end, some of the tiles will pass the test and will be given a value. Others (that had signal in them) will just be assigned a NaN (not-a-number) value. But we need a measurement over each tile (and thus pixel). We will therefore use interpolation to assign a value to the NaN tiles.

However, prior to interpolating over the failed tiles, another point should be considered: large and extended galaxies, or bright stars, have wings which sink into the noise very gradually. In some cases, the gradient over these wings can be on scales that is larger than the tiles (for example the pixel value changes by 0.1$\sigma$ after 100 pixels, but the tile has a width of 30 pixels).

In such cases, the $q_{\text{mean}}$ test will be successful, even though there is signal. Recall that $q_{\text{mean}}$ is a measure of skewness. If we don’t identify (and thus set to NaN) such outlier tiles before the interpolation, the photons of the outskirts of the objects will leak into the detection thresholds or Sky and Sky STD measurements and bias our result, see Section 2.3 [Detecting large extended targets], page 76. Therefore, the final step of “quantifying signal in a tile” is to look at this distribution of successful tiles and remove the outliers. $\sigma$-clipping is a good solution for removing a few outliers, but the problem with outliers of this kind is that there may be many such tiles (depending on the large/bright stars/galaxies in the image). We therefore apply the following local outlier rejection strategy.

For each tile, we find the nearest $N_{\text{ngb}}$ tiles that had a usable value ($N_{\text{ngb}}$ is the value given to --interpnumngb). We then sort them and find the difference between the largest and second-to-smallest elements (The minimum isn’t used because the scatter can be large). Let’s call this the tile’s slope (measured from its neighbors). All the tiles that are on a region of flat noise will have similar slope values, but if a few tiles fall on the wings of a bright star
or large galaxy, their slope will be significantly larger than the tiles with no signal. We just have to find the smallest tile slope value that is an outlier compared to the rest, and reject all tiles with a slope larger than that.

To identify the smallest outlier, we’ll use the distribution of distances between sorted elements. Let’s assume the total number tiles with a good mean-median quantile difference is \( N \). We sort them in increasing order based on their slope (defined above). So the array of tile slopes can be written as \( \{s[0], s[1], ..., s[N]\} \), where \( s[0] < s[1] \) and so on. We then start from element \( M \) and calculate the distances between all two adjacent values before it: \( \{s[1] - s[0], s[2] - s[1], s[M - 1] - s[M - 2]\} \).

The \( \sigma \)-clipped median and standard deviation of this distribution are then found (this \( \sigma \)-clipping is configured with --outliersclip which takes two values, see Section 7.1.3 [Sigma clipping], page 343). If the distance between the element and its previous element is more than --outliersigma multiples of the \( \sigma \)-clipped standard deviation added with the \( \sigma \)-clipped median, that element is considered an outlier and all tiles larger than that value are ignored.

Formally, if we assume there are \( N \) elements. They are first sorted. Searching for the outlier starts on element \( N/3 \) (integer division). Let’s take \( v_i \) to be the \( i \)-th element of the sorted input (with no blank values) and \( m \) and \( \sigma \) as the \( \sigma \)-clipped median and standard deviation from the distances of the previous \( N/3 - 1 \) elements (not including \( v_i \)). If the value given to --outliersigma is displayed with \( s \), the \( i \)-th element is considered as an outlier when the condition below is true.

\[
\frac{(v_i - v_{i-1}) - m}{\sigma} > s
\]

Since \( i \) begins from the \( N/3 \)-th element in the sorted array (a quantile of \( 1/3 = 0.33 \)), the outlier has to be larger than the 0.33 quantile value of the dataset.

Once the outlying tiles have been successfully identified and set to NaN, we use nearest-neighbor interpolation to give a value to all tiles in the image. We don’t use parametric interpolation methods (like bicubic), because they will effectively extrapolate on the edges, creating strong artifacts. Nearest-neighbor interpolation is very simple: for each tile, we find the \( N_{ngb} \) nearest tiles that had a good value, the tile’s value is found by estimating the median. You can set \( N_{ngb} \) through the --interpnumngb option. Once all the tiles are given a value, a smoothing step is implemented to remove the sharp value contrast that can happen on the edges of tiles. The size of the smoothing box is set with the --smoothwidth option.

As mentioned above, the process above is used for any of the basic measurements (for example identifying the quantile-based thresholds in NoiseChisel, or the Sky value in Statistics). You can use the check-image feature of NoiseChisel or Statistics to inspect the steps and visually see each step (all the options that start with --check). For example, as mentioned in the Section 2.3.2 [NoiseChisel optimization], page 78, tutorial, when given a dataset from a new instrument (with differing noise properties), we highly recommend to use --checkqthresh in your first call and visually inspect how the parameters above affect the final quantile threshold (e.g., have the wings of bright sources leaked into the threshold?). The same goes for the --checksky option of Statistics or NoiseChisel.
7.1.5 Invoking Statistics

Statistics will print statistical measures of an input dataset (table column or image). The executable name is `aststatistics` with the following general template

```
$ aststatistics [OPTION ...] InputImage.fits
```

One line examples:

```
## Print some general statistics of input image:
$ aststatistics image.fits

## Print some general statistics of column named MAG_F160W:
$ aststatistics catalog.fits -h1 --column=MAG_F160W

## Make the histogram of the column named MAG_F160W:
$ aststatistics table.fits -cMAG_F160W --histogram

## Find the Sky value on image with a given kernel:
$ aststatistics image.fits --sky --kernel=kernel.fits

## Print Sigma-clipped results of records with a MAG_F160W
## column value between 26 and 27:
$ aststatistics cat.fits -cMAG_F160W -g26 -l27 --sigmaclip=3,0.2

## Print the median value of all records in column MAG_F160W that
## have a value larger than 3 in column PHOTO_Z:
$ aststatistics tab.txt -rPHOTO_Z -g3 -cMAG_F160W --median

## Calculate the median of the third column in the input table, but only
## for rows where the mean of the first and second columns is >5.
$ awk '($1+$2)/2 > 5 {print $3}' table.txt | aststatistics --median
```

Statistics can take its input dataset either from a file (image or table) or the Standard input (see Section 4.1.4 [Standard input], page 167). If any output file is to be created, the value to the `--output` option, is used as the base name for the generated files. Without `--output`, the input name will be used to generate an output name, see Section 4.9 [Automatic output], page 191. The options described below are particular to Statistics, but for general operations, it shares a large collection of options with the other Gnuastro programs, see Section 4.1.2 [Common options], page 156, for the full list. For more on reading from standard input, please see the description of `--stdintimeout` option in Section 4.1.2.1 [Input/Output options], page 157. Options can also be given in configuration files, for more, please see Section 4.2 [Configuration files], page 169.

The input dataset may have blank values (see Section 6.1.3 [Blank pixels], page 262), in this case, all blank pixels are ignored during the calculation. Initially, the full dataset will be read, but it is possible to select a specific range of data elements to use in the analysis of each run. You can either directly specify a minimum and maximum value for the range of data elements to use (with `--greaterequal` or `--lessthan`), or specify the range using quantiles (with `--qrange`). If a range is specified, all pixels outside of it are ignored before any processing.
The following set of options are for specifying the input/outputs of Statistics. There are many other input/output options that are common to all Gnuastro programs including Statistics, see Section 4.1.2.1 [Input/Output options], page 157, for those.

-c STR/INT
--column=STR/INT
The column to use when the input file is a table with more than one column. See Section 4.7.3 [Selecting table columns], page 188, for a full description of how to use this option. For more on how tables are read in Gnuastro, please see Section 4.7 [Tables], page 183.

-r STR/INT
--refcol=STR/INT
The reference column selector when the input file is a table. When a reference column is given, the range options below will be applied to this column and only elements in the input column that have a reference value in the correct range will be used. In practice this option allows you to select a subset of the input column based on values in another (the reference) column. All the statistical calculations will be done on the selected input column, not the reference column.

-g FLT
--greaterequal=FLT
Limit the range of inputs into those with values greater and equal to what is given to this option. None of the values below this value will be used in any of the processing steps below.

-l FLT
--lessthan=FLT
Limit the range of inputs into those with values less-than what is given to this option. None of the values greater or equal to this value will be used in any of the processing steps below.

-Q FLT[,FLT]
--qrange=FLT[,FLT]
Specify the range of usable inputs using the quantile. This option can take one or two quantiles to specify the range. When only one number is input (let’s call it \(Q\)), the range will be those values in the quantile range \(Q\) to \(1 - Q\). So when only one value is given, it must be less than 0.5. When two values are given, the first is used as the lower quantile range and the second is used as the larger quantile range.

The quantile of a given element in a dataset is defined by the fraction of its index to the total number of values in the sorted input array. So the smallest and largest values in the dataset have a quantile of 0.0 and 1.0. The quantile is a very useful non-parametric (making no assumptions about the input) relative measure to specify a range. It can best be understood in terms of the cumulative frequency plot, see Section 7.1.1 [Histogram and Cumulative Frequency Plot], page 337. The quantile of each horizontal axis value in the cumulative frequency plot is the vertical axis value associate with it.

When no operation is requested, Statistics will print some general basic properties of the input dataset on the command-line like the example below (ran on one of the output
images of make check⁶). This default behavior is designed to help give you a general feeling of how the data are distributed and help in narrowing down your analysis.

```
$ aststatistics convolve_spatial_scaled_noised.fits
   --greaterequal=9500 --lessthan=11000
Statistics (GNU Astronomy Utilities) X.X
-------
Input: convolve_spatial_scaled_noised.fits (hdu: 0)
Range: from (inclusive) 9500, upto (exclusive) 11000.
Unit: Brightness
-------
Number of elements: 9074
Minimum: 9622.35
Maximum: 10999.7
Mode: 10055.45996
Mode quantile: 0.4001983908
Median: 10093.7
Mean: 10143.98257
Standard deviation: 221.80834
-------
Histogram:
|**
|*****
|*******
|*********
|***********
|*************
|**************
|***************
|***************
|***************
|***************
|***************
|***************
|***************
|***************
|***************
|***************
|***************
|***************
|***************

Gnuastro's Statistics is a very general purpose program, so to be able to easily understand this diversity in its operations (and how to possibly run them together), we'll divided the operations into two types: those that don't respect the position of the elements and those that do (by tessellating the input on a tile grid, see Section 4.8 [Tessellation], page 189). The former treat the whole dataset as one and can re-arrange all the elements (for example sort them), but the former do their processing on each tile independently. First, we'll review the operations that work on the whole dataset.

The group of options below can be used to get single value measurement(s) of the whole dataset. They will print only the requested value as one field in a line/row, like the --mean, --median options. These options can be called any number of times and in any order. The outputs of all such options will be printed on one line following each other (with a space character between them). This feature makes these options very useful in scripts, or to

---

⁶ You can try it by running the command in the tests directory, open the image with a FITS viewer and have a look at it to get a sense of how these statistics relate to the input image/dataset.
redirect into programs like GNU AWK for higher-level processing. These are some of the most basic measures, Gnuastro is still under heavy development and this list will grow. If you want another statistical parameter, please contact us and we will do out best to add it to this list, see Section 1.9 [Suggest new feature], page 13.

-n
   --number  Print the number of all used (non-blank and in range) elements.

   --minimum
   Print the minimum value of all used elements.

   --maximum
   Print the maximum value of all used elements.

   --sum
   Print the sum of all used elements.

-m
   --mean
   Print the mean (average) of all used elements.

-t
   --std
   Print the standard deviation of all used elements.

-E
   --median
   Print the median of all used elements.

-u FLT[,FLT[,...]]
   --quantile=FLT[,FLT[,...]]
   Print the values at the given quantiles of the input dataset. Any number of quantiles may be given and one number will be printed for each. Values can either be written as a single number or as fractions, but must be between zero and one (inclusive). Hence, in effect --quantile=0.25 --quantile=0.75 is equivalent to --quantile=0.25,3/4, or -u1/4,3/4.

   The returned value is one of the elements from the dataset. Taking q to be your desired quantile, and N to be the total number of used (non-blank and within the given range) elements, the returned value is at the following position in the sorted array: round(q × N).

   --quantfunc=FLT[,FLT[,...]]
   Print the quantiles of the given values in the dataset. This option is the inverse of the --quantile and operates similarly except that the acceptable values are within the range of the dataset, not between 0 and 1. Formally it is known as the “Quantile function”.

   Since the dataset is not continuous this function will find the nearest element of the dataset and use its position to estimate the quantile function.

   --quantofmean
   Print the quantile of the mean in the dataset. This is a very good measure of detecting skewness or outliers. The concept is used by programs like NoiseChisel to identify the presence of signal in a tile of the image (because signal in noise causes skewness).

   For example, take this simple array: 1 2 20 4 5 6 3. The mean is 5.85. The nearest element to this mean is 6 and the quantile of 6 in this distribution is
Here is how we got to this: in the sorted dataset (1 2 3 4 5 6 20), 6 is the 5-th element (counting from zero, since a quantile of zero corresponds to the minimum, by definition) and the maximum is the 6-th element (again, counting from zero). So the quantile of the mean in this case is 5/6 = 0.8333.

In the example above, if we had 7 instead of 20 (which was an outlier), then the mean would be 4 and the quantile of the mean would be 0.5 (which by definition, is the quantile of the median), showing no outliers. As the number of elements increases, the mean itself is less affected by a small number of outliers, but skewness can be nicely identified by the quantile of the mean.

- **--mode**
  Print the mode of all used elements. The mode is found through the mirror distribution which is fully described in Appendix C of Akhlaghi and Ichikawa 2015 (https://arxiv.org/abs/1505.01664). See that section for a full description. This mode calculation algorithm is non-parametric, so when the dataset is not large enough (larger than about 1000 elements usually), or doesn’t have a clear mode it can fail. In such cases, this option will return a value of `nan` (for the floating point NaN value).

  As described in that paper, the easiest way to assess the quality of this mode calculation method is to use it’s symmetricity (see `--modesym` below). A better way would be to use the `--mirror` option to generate the histogram and cumulative frequency tables for any given mirror value (the mode in this case) as a table. If you generate plots like those shown in Figure 21 of that paper, then your mode is accurate.

- **--modequant**
  Print the quantile of the mode. You can get the actual mode value from the `--mode` described above. In many cases, the absolute value of the mode is irrelevant, but its position within the distribution is important. In such cases, this option will become handy.

- **--modesym**
  Print the symmetricity of the calculated mode. See the description of `--mode` for more. This mode algorithm finds the mode based on how symmetric it is, so if the symmetricity returned by this option is too low, the mode is not too accurate. See Appendix C of Akhlaghi and Ichikawa 2015 (https://arxiv.org/abs/1505.01664) for a full description. In practice, symmetricity values larger than 0.2 are mostly good.

- **--modesymvalue**
  Print the value in the distribution where the mirror and input distributions are no longer symmetric, see `--mode` and Appendix C of Akhlaghi and Ichikawa 2015 (https://arxiv.org/abs/1505.01664) for more.

- **--sigclip-number**
  Number of elements after applying $\sigma$-clipping (see Section 7.1.3 [Sigma clipping], page 343). $\sigma$-clipping configuration is done with the `--sigclippars` option.
--sigclip-median
Median after applying $\sigma$-clipping (see Section 7.1.3 [Sigma clipping], page 343). $\sigma$-clipping configuration is done with the \(--\text{sigclipparams}\) option.

Here is one scenario where this can be useful: assume you have a table and you would like to remove the rows that are outliers (not within the $\sigma$-clipping range). Let’s assume your table is called \texttt{table.fits} and you only want to keep the rows that have a value in \texttt{COLUMN} within the $\sigma$-clipped range (to $3\sigma$, with a tolerance of 0.1). This command will return the $\sigma$-clipped median and standard deviation (used to define the range later).

\[
\texttt{\$ aststatistics table.fits } -c\text{COLUMN} \text{ --sclipparams=3,0.1} \\
\texttt{--sigclip-median --sigclip-std}
\]

You can then use the \(--\text{range}\) option of Table (see Section 5.3 [Table], page 230) to select the proper rows. But for that, you need the actual starting and ending values of the range ($m \pm s\sigma$; where $m$ is the median and $s$ is the multiple of sigma to define an outlier). Therefore, the raw outputs of Statistics in the command above aren’t enough.

To get the starting and ending values of the non-outlier range (and put a ‘,’ between them, ready to be used in \(--\text{range}\)), pipe the result into AWK. But in AWK, we’ll also need the multiple of $\sigma$, so we’ll define it as a shell variable ($s$) before calling Statistics (note how $s$ is used two times now):

\[
\texttt{\$ s=3} \\
\texttt{\$ aststatistics table.fits } -c\text{COLUMN} \text{ --sclipparams=$s$,0.1} \\
\texttt{--sigclip-median --sigclip-std} \\
\texttt{\textbar \textbar \text{"\texttt{awk '{s='$s'; printf("%f,%f\n", $1-s*$2, $1+s*$2)}'}')}\]

To pass it onto Table, we’ll need to keep the printed output from the command above in another shell variable ($r$), not print it. In Bash, can do this by putting the whole statement within a $()$:

\[
\texttt{\$ s=3} \\
\texttt{\$ r=$(aststatistics table.fits } -c\text{COLUMN} \text{ --sclipparams=$s$,0.1} \\
\texttt{--sigclip-median --sigclip-std} \\
\texttt{\textbar \textbar \text{"\texttt{awk '{s='$s'; printf("%f,%f\n", $1-s*$2, $1+s*$2)}'}')}\]
\texttt{\$ echo $r} \
\texttt{\text{"\texttt{# Just to confirm.}}\}
\]

Now you can use Table with the \(--\text{range}\) option to only print the rows that have a value in \texttt{COLUMN} within the desired range:

\[
\texttt{\$ asttable table.fits --range=COLUMN,$r}
\]

To save the resulting table (that is clean of outliers) in another file (for example named \texttt{cleaned.fits}, it can also have a .txt suffix), just add \(--\text{output=cleaned.fits}\) to the command above.

--sigclip-mean
Mean after applying $\sigma$-clipping (see Section 7.1.3 [Sigma clipping], page 343). $\sigma$-clipping configuration is done with the \(--\text{sigclipparams}\) option.

--sigclip-std
Standard deviation after applying $\sigma$-clipping (see Section 7.1.3 [Sigma clipping], page 343). $\sigma$-clipping configuration is done with the \(--\text{sigclipparams}\) option.
The list of options below are for those statistical operations that output more than one value. So while they can be called together in one run, their outputs will be distinct (each one’s output will usually be printed in more than one line).

- A
  --ascihist

  Print an ASCII histogram of the usable values within the input dataset along with some basic information like the example below (from the UVUDF catalog7). The width and height of the histogram (in units of character widths and heights on your command-line terminal) can be set with the --numasciibins (for the width) and --asciiheight options.

  For a full description of the histogram, please see Section 7.1.1 [Histogram and Cumulative Frequency Plot], page 337. An ASCII plot is certainly very crude and cannot be used in any publication, but it is very useful for getting a general feeling of the input dataset very fast and easily on the command-line without having to take your hands off the keyboard (which is a major distraction!). If you want to try it out, you can write it all in one line and ignore the \ and extra spaces.

  $ aststatistics uvudf_rafelski_2015.fits.gz --hdu=1 \ 
  --column=MAG_F160W --lessthan=40 \ 
  --ascihist --numasciibins=55

  ASCII Histogram:
  Number: 8593
  Y: (linear: 0 to 660)
  X: (linear: 17.7735 -- 31.4679, in 55 bins)

  ****
  *****
  ******
  ********
  *********
  **********
  ***********
  ************
  *************
  **************
  ***************
  *****************
  ******************
  *******************
  ****************************
  ****************************
  ****************************
  ****************************
  ****************************
  ****************************
  ****************************
  ****************************
  ****************************
  ****************************
  ****************************

  --asciicfp

  Print the cumulative frequency plot of the usable elements in the input dataset. Please see descriptions under --ascihist for more, the example below is from the same input table as that example. To better understand the cumulative frequency plot, please see Section 7.1.1 [Histogram and Cumulative Frequency Plot], page 337.

  $ aststatistics uvudf_rafelski_2015.fits.gz --hdu=1 \ 
  --column=MAG_F160W --lessthan=40 \ 

7 https://asd.gsfc.nasa.gov/UVUDF/uvudf_rafelski_2015.fits.gz
--asciicfp --numasciibins=55

ASCII Cumulative frequency plot:
Y: (linear: 0 to 8593)
X: (linear: 17.7735 -- 31.4679, in 55 bins)
| *******
| ********
| *********
| **********
| ***********
| ************
| *************
| **************
| ****************
| *****************
| *******************
| **********************
| ********************************

-H
--histogram

Save the histogram of the usable values in the input dataset into a table. The first column is the value at the center of the bin and the second is the number of points in that bin. If the --cumulative option is also called with this option in a run, then the table will have three columns (the third is the cumulative frequency plot). Through the --numbins, --onebinstart, or --manualbinrange, you can modify the first column values and with --normalize and --maxbinone you can modify the second columns. See below for the description of each.

By default (when no --output is specified) a plain text table will be created, see Section 4.7.2 [Gnuastro text table format], page 185. If a FITS name is specified, you can use the common option --tableformat to have it as a FITS ASCII or FITS binary format, see Section 4.1.2 [Common options], page 156. This table can then be fed into your favorite plotting tool and get a much more clean and nice histogram than what the raw command-line can offer you (with the --asciihist option).

--histogram2d

Save the 2D histogram of two input columns into an output file, see Section 7.1.2 [2D Histograms], page 338. The output will have three columns: the first two are the coordinates of each box’s center in the first and second dimensions/columns. The third will be number of input points that fall within that box.

-C
--cumulative

Save the cumulative frequency plot of the usable values in the input dataset into a table, similar to --histogram.
-s
--sigmaclip
Do $\sigma$-clipping on the usable pixels of the input dataset. See Section 7.1.3 [Sigma clipping], page 343, for a full description on $\sigma$-clipping and also to better understand this option. The $\sigma$-clipping parameters can be set through the --sclipparams option (see below).

--mirror=FLT
Make a histogram and cumulative frequency plot of the mirror distribution for the given dataset when the mirror is located at the value to this option. The mirror distribution is fully described in Appendix C of Akhlaghi and Ichikawa 2015 (https://arxiv.org/abs/1505.01664) and currently it is only used to calculate the mode (see --mode).

Just note that the mirror distribution is a discrete distribution like the input, so while you may give any number as the value to this option, the actual mirror value is the closest number in the input dataset to this value. If the two numbers are different, Statistics will warn you of the actual mirror value used.

This option will make a table as output. Depending on your selected name for the output, it will be either a FITS table or a plain text table (which is the default). It contains three columns: the first is the center of the bins, the second is the histogram (with the largest value set to 1) and the third is the normalized cumulative frequency plot of the mirror distribution. The bins will be positioned such that the mode is on the starting interval of one of the bins to make it symmetric around the mirror. With this output file and the input histogram (that you can generate in another run of Statistics, using the --onebinvalue), it is possible to make plots like Figure 21 of Akhlaghi and Ichikawa 2015 (https://arxiv.org/abs/1505.01664).

The list of options below allow customization of the histogram and cumulative frequency plots (for the --histogram, --cumulative, --asciihist, and --asciicfp options).

--numbins
The number of bins (rows) to use in the histogram and the cumulative frequency plot tables (outputs of --histogram and --cumulative).

--numasciibins
The number of bins (characters) to use in the ASCII plots when printing the histogram and the cumulative frequency plot (outputs of --asciihist and --asciicfp).

--asciheight
The number of lines to use when printing the ASCII histogram and cumulative frequency plot on the command-line (outputs of --asciihist and --asciicfp).

-n
--normalize
Normalize the histogram or cumulative frequency plot tables (outputs of --histogram and --cumulative). For a histogram, the sum of all bins will become one and for a cumulative frequency plot the last bin value will be one.
--maxbinone
Divide all the histogram values by the maximum bin value so it becomes one and the rest are similarly scaled. In some situations (for example if you want to plot the histogram and cumulative frequency plot in one plot) this can be very useful.

--onebinstart=FLT
Make sure that one bin starts with the value to this option. In practice, this will shift the bins used to find the histogram and cumulative frequency plot such that one bin’s lower interval becomes this value.

For example when a histogram range includes negative and positive values and zero has a special significance in your analysis, then zero might fall somewhere in one bin. As a result that bin will have counts of positive and negative. By setting --onebinstart=0, you can make sure that one bin will only count negative values in the vicinity of zero and the next bin will only count positive ones in that vicinity.

Note that by default, the first row of the histogram and cumulative frequency plot show the central values of each bin. So in the example above you will not see the 0.000 in the first column, you will see two symmetric values.

If the value is not within the usable input range, this option will be ignored. When it is, this option is the last operation before the bins are finalized, therefore it has a higher priority than options like --manualbinrange.

--manualbinrange
Use the values given to the --greaterequal and --lessthan to define the range of all bin-based calculations like the histogram. This option itself doesn’t take any value, but just tells the program to use the values of those two options instead of the minimum and maximum values of a plot. If any of the two options are not given, then the minimum or maximum will be used respectively. Therefore, if none of them are called calling this option is redundant.

The --onebinstart option has a higher priority than this option. In other words, --onebinstart takes effect after the range has been finalized and the initial bins have been defined, therefore it has the power to (possibly) shift the bins. If you want to manually set the range of the bins and have one bin on a special value, it is thus better to avoid --onebinstart.

--numbins2=INT
Similar to --numbins, but for the second column when a 2D histogram is requested, see --histogram2d.

--greaterequal2=FLT
Similar to --greaterequal, but for the second column when a 2D histogram is requested, see --histogram2d.

--lessthan2=FLT
Similar to --lessthan, but for the second column when a 2D histogram is requested, see --histogram2d.
--onebinstart2=FLT

Similar to --onebinstart, but for the second column when a 2D histogram is requested, see --histogram2d.

All the options described until now were from the first class of operations discussed above: those that treat the whole dataset as one. However, it often happens that the relative position of the dataset elements over the dataset is significant. For example you don’t want one median value for the whole input image, you want to know how the median changes over the image. For such operations, the input has to be tessellated (see Section 4.8 [Tessellation], page 189). Thus this class of options can’t currently be called along with the options above in one run of Statistics.

-t
--ontile

Do the respective single-valued calculation over one tile of the input dataset, not the whole dataset. This option must be called with at least one of the single valued options discussed above (for example --mean or --quantile). The output will be a file in the same format as the input. If the --oneelempertile option is called, then one element/pixel will be used for each tile (see Section 4.1.2.2 [Processing options], page 159). Otherwise, the output will have the same size as the input, but each element will have the value corresponding to that tile’s value. If multiple single valued operations are called, then for each operation there will be one extension in the output FITS file.

-R FLT[,FLT[,FLT...]]
--contour=FLT[,FLT[,FLT...]]

Write the contours for the requested levels in a file ending with _contour.txt. It will have three columns: the first two are the coordinates of each point and the third is the level it belongs to (one of the input values). Each disconnected contour region will be separated by a blank line. This is the requested format for adding contours with PGFPLOTS in \LaTeX. If any other format can be useful for your work please let us know so we can add it. If the image has World Coordinate System information, the written coordinates will be in RA and Dec, otherwise, they will be in pixel coordinates.

Note that currently, this is a very crude/simple implementation, please let us know if you find problematic situations so we can fix it.

-y
--sky

Estimate the Sky value on each tile as fully described in Section 7.1.4.3 [Quantifying signal in a tile], page 347. As described in that section, several options are necessary to configure the Sky estimation which are listed below. The output file will have two extensions: the first is the Sky value and the second is the Sky standard deviation on each tile. Similar to --ontile, if the --oneelempertile option is called, then one element/pixel will be used for each tile (see Section 4.1.2.2 [Processing options], page 159).

The parameters for estimating the sky value can be set with the following options, except for the --scilpparams option (which is also used by the --sigmaclip), the rest are only used for the Sky value estimation.
-k=FITS
--kernel=FITS
File name of kernel to help in estimating the significance of signal in a tile, see Section 7.1.4.3 [Quantifying signal in a tile], page 347.

--khdu=STR
Kernel HDU to help in estimating the significance of signal in a tile, see Section 7.1.4.3 [Quantifying signal in a tile], page 347.

--meanmedqdiff=FLT
The maximum acceptable distance between the quantiles of the mean and median, see Section 7.1.4.3 [Quantifying signal in a tile], page 347. The initial Sky and its standard deviation estimates are measured on tiles where the quantiles of their mean and median are less distant than the value given to this option. For example --meanmedqdiff=0.01 means that only tiles where the mean’s quantile is between 0.49 and 0.51 (recall that the median’s quantile is 0.5) will be used.

--sclipparams=FLT,FLT
The $\sigma$-clipping parameters, see Section 7.1.3 [Sigma clipping], page 343. This option takes two values which are separated by a comma (.). Each value can either be written as a single number or as a fraction of two numbers (for example $3,1/10$). The first value to this option is the multiple of $\sigma$ that will be clipped ($\alpha$ in that section). The second value is the exit criteria. If it is less than 1, then it is interpreted as tolerance and if it is larger than one it is a specific number. Hence, in the latter case the value must be an integer.

--outliersclip=FLT,FLT
$\sigma$-clipping parameters for the outlier rejection of the Sky value (similar to --sclipparams).

Outlier rejection is useful when the dataset contains a large and diffuse (almost flat within each tile) signal. The flatness of the profile will cause it to successfully pass the mean-median quantile difference test, so we’ll need to use the distribution of successful tiles for removing these false positive. For more, see the latter half of Section 7.1.4.3 [Quantifying signal in a tile], page 347.

--outliersigma=FLT
Multiple of sigma to define an outlier in the Sky value estimation. If this option is given a value of zero, no outlier rejection will take place. For more see --outliersclip and the latter half of Section 7.1.4.3 [Quantifying signal in a tile], page 347.

--smoothwidth=INT
Width of a flat kernel to convolve the interpolated tile values. Tile interpolation is done using the median of the --interpnumngb neighbors of each tile (see Section 4.1.2.2 [Processing options], page 159). If this option is given a value of zero or one, no smoothing will be done. Without smoothing, strong boundaries will probably be created between the values estimated for each tile. It is thus good to smooth the interpolated image so strong discontinuities do not show up in the final Sky values. The smoothing is done through convolution (see
Section 6.3.1.1 [Convolution process], page 307) with a flat kernel, so the value to this option must be an odd number.

`--ignoreblankintiles`

Don’t set the input’s blank pixels to blank in the tiled outputs (for example Sky and Sky standard deviation extensions of the output). This is only applicable when the tiled output has the same size as the input, in other words, when `--oneelempertile` isn’t called.

By default, blank values in the input (commonly on the edges which are outside the survey/field area) will be set to blank in the tiled outputs also. But in other scenarios this default behavior is not desired: for example if you have masked something in the input, but want the tiled output under that also.

`--checksky`

Create a multi-extension FITS file showing the steps that were used to estimate the Sky value over the input, see Section 7.1.4.3 [Quantifying signal in a tile], page 347. The file will have two extensions for each step (one for the Sky and one for the Sky standard deviation).

### 7.2 NoiseChisel

Once instrumental signatures are removed from the raw data (image) in the initial reduction process (see Chapter 6 [Data manipulation], page 259). You are naturally eager to start answering the scientific questions that motivated the data collection in the first place. However, the raw dataset/image is just an array of values/pixels, that is all! These raw values cannot directly be used to answer your scientific questions: for example “how many galaxies are there in the image?”, “What is their brightness?” and etc.

The first high-level step your analysis will therefore be to classify, or label, the dataset elements (pixels) into two classes: 1) Noise, where random effects are the major contributor to the value, and 2) Signal, where non-random factors (for example light from a distant galaxy) are present. This classification of the elements in a dataset is formally known as detection.

In an observational/experimental dataset, signal is always buried in noise: only mock/simulated datasets are free of noise. Therefore detection, or the process of separating signal from noise, determines the number of objects you study and the accuracy of any higher-level measurement you do on them. Detection is thus the most important step of any analysis and is not trivial. In particular, the most scientifically interesting astronomical targets are faint, can have a large variety of morphologies, along with a large distribution in brightness and size. Therefore when noise is significant, proper detection of your targets is a uniquely decisive step in your final scientific analysis/result.

NoiseChisel is Gnuastro’s program for detection of targets that don’t have a sharp border (almost all astronomical objects). When the targets have sharp edges/borders (for example cells in biological imaging), a simple threshold is enough to separate them from noise and each other (if they are not touching). To detect such sharp-edged targets, you can use Gnuastro’s Arithmetic program in a command like below (assuming the threshold is 100, see Section 6.2 [Arithmetic], page 272):

```
$ astarithmetic in.fits 100 gt 2 connected-components
```
Since almost no astronomical target has such sharp edges, we need a more advanced detection methodology. NoiseChisel uses a new noise-based paradigm for detection of very extended and diffuse targets that are drowned deeply in the ocean of noise. It was initially introduced in Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664) and improvements after the first four were published in Akhlaghi [2019] (https://arxiv.org/abs/1909.11230). Please take the time to go through these papers to most effectively understand the need of NoiseChisel and how best to use it.

The name of NoiseChisel is derived from the first thing it does after thresholding the dataset: to erode it. In mathematical morphology, erosion on pixels can be pictured as carving-off boundary pixels. Hence, what NoiseChisel does is similar to what a wood chisel or stone chisel do. It is just not a hardware, but a software. In fact, looking at it as a chisel and your dataset as a solid cube of rock will greatly help in effectively understanding and optimally using it: with NoiseChisel you literally carve your targets out of the noise. Try running it with the \texttt{--checkdetection} option, and open the temporary output as a multi-extension cube, to see each step of the carving process on your input dataset (see Section 10.4 [Viewing FITS file contents with DS9 or TOPCAT], page 494).

NoiseChisel’s primary output is a binary detection map with the same size as the input but its pixels only have two values: 0 (background) and 1 (foreground). Pixels that don’t harbor any detected signal (noise) are given a label (or value) of zero and those with a value of 1 have been identified as hosting signal.

Segmentation is the process of classifying the signal into higher-level constructs. For example if you have two separate galaxies in one image, NoiseChisel will give a value of 1 to the pixels of both (each forming an “island” of touching foreground pixels). After segmentation, the connected foreground pixels will get separate labels, enabling you to study them individually. NoiseChisel is only focused on detection (separating signal from noise), to segment the signal (into separate galaxies for example). Gnuastro has a separate specialized program Section 7.3 [Segment], page 381. NoiseChisel’s output can be directly/readily fed into Segment.

For more on NoiseChisel’s output format and its benefits (especially in conjunction with Section 7.3 [Segment], page 381, and later Section 7.4 [MakeCatalog], page 392), please see Akhlaghi [2016] (https://arxiv.org/abs/1611.06387). Just note that when that paper was published, Segment was not yet spun-off into a separate program, and NoiseChisel done both detection and segmentation.

NoiseChisel’s output is designed to be generic enough to be easily used in any higher-level analysis. If your targets are not touching after running NoiseChisel and you aren’t interested in their sub-structure, you don’t need the Segment program at all. You can ask NoiseChisel to find the connected pixels in the output with the \texttt{--label} option. In this case, the output won’t be a binary image any more, the signal will have counters/labels starting from 1 for each connected group of pixels. You can then directly feed NoiseChisel’s output into MakeCatalog for measurements over the detections and the production of a catalog (see Section 7.4 [MakeCatalog], page 392).

Thanks to the published papers mentioned above, there is no need to provide a more complete introduction to NoiseChisel in this book. However, published papers cannot be updated any more, but the software has evolved/changed. The changes since publication are documented in Section 7.2.1 [NoiseChisel changes after publication], page 364. In
Section 7.2.2 [Invoking NoiseChisel], page 365, the details of running NoiseChisel and its options are discussed.

As discussed above, detection is one of the most important steps for your scientific result. It is therefore very important to obtain a good understanding of NoiseChisel (and afterwards Section 7.3 [Segment], page 381, and Section 7.4 [MakeCatalog], page 392). We strongly recommend reviewing two tutorials of Section 2.2 [General program usage tutorial], page 29, and Section 2.3 [Detecting large extended targets], page 76. They are designed to show how to most effectively use NoiseChisel for the detection of small faint objects and large extended objects. In the meantime, they also show the modular principle behind Gnuastro’s programs and how they are built to complement, and build upon, each other.

Section 2.2 [General program usage tutorial], page 29, culminates in using NoiseChisel to detect galaxies and use its outputs to find the galaxy colors. Defining colors is a very common process in most science-cases. Therefore it is also recommended to (patiently) complete that tutorial for optimal usage of NoiseChisel in conjunction with all the other Gnuastro programs. Section 2.3 [Detecting large extended targets], page 76, shows you can optimize NoiseChisel’s settings for very extended objects to successfully carve out to signal-to-noise ratio levels of below 1/10. After going through those tutorials, play a little with the settings (in the order presented in the paper and Section 7.2.2 [Invoking NoiseChisel], page 365) on a dataset you are familiar with and inspect all the check images (options starting with --check) to see the effect of each parameter.

Below, in Section 7.2.2 [Invoking NoiseChisel], page 365, we will review NoiseChisel’s input, detection, and output options in Section 7.2.2.1 [NoiseChisel input], page 367, Section 7.2.2.2 [Detection options], page 370, and Section 7.2.2.3 [NoiseChisel output], page 378. If you have used NoiseChisel within your research, please run it with --cite to list the papers you should cite and how to acknowledge its funding sources.

### 7.2.1 NoiseChisel changes after publication

NoiseChisel was initially introduced in Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664) and updates after the first four years were published in Akhlaghi [2019] (https://arxiv.org/abs/1909.11230). To help in understanding how it works, those papers have many figures showing every step on multiple mock and real examples. We recommended to read these papers for a good understanding of what it does and how each parameter influences the output.

However, the papers cannot be updated anymore, but NoiseChisel has evolved (and will continue to do so): better algorithms or steps have been found and implemented and some options have been added, removed or changed behavior. This book is thus the final and definitive guide to NoiseChisel. The aim of this section is to make the transition from the papers above to the installed version on your system, as smooth as possible with the list below. For a more detailed list of changes in each Gnuastro version, please see the NEWS file\(^8\).

- An improved outlier rejection for identifying tiles without any signal has been implemented in the quantile-threshold phase: Prior to version 0.14, outliers were defined globally: the distribution of all tiles with an acceptable --meanmedqdiff was inspected and outliers were found and rejected. However, this caused problems when there are

---

\(^8\) The NEWS file is present in the released Gnuastro tarball, see Section 3.2.1 [Release tarball], page 131.
strong gradients over the image (for example an image prior to flat-fielding, or in the presence of a large foreground galaxy). In these cases, the faint wings of galaxies/stars could be mistakenly identified as Sky (leaving a footprint of the object on the Sky output) and wrongly subtracted.

It was possible to play with the parameters to correct this for that particular dataset, but that was frustrating. Therefore from version 0.14, instead of finding outliers from the full tile distribution, we now measure the slope of the tile’s nearby tiles and find outliers locally. For more on the outlier-by-distance algorithm and the definition of slope, see Section 7.1.4.3 [Quantifying signal in a tile], page 347. In our tests, this gave a much improved estimate of the quantile thresholds and final Sky values with default values.

7.2.2 Invoking NoiseChisel

NoiseChisel will detect signal in noise producing a multi-extension dataset containing a binary detection map which is the same size as the input. Its output can be readily used for input into Section 7.3 [Segment], page 381, for higher-level segmentation, or Section 7.4 [MakeCatalog], page 392, to do measurements and generate a catalog. The executable name is astnoisechisel with the following general template

$ astnoisechisel [OPTION ...] InputImage.fits

One line examples:

## Detect signal in input.fits.
$ astnoisechisel input.fits

## Inspect all the detection steps after changing a parameter.
$ astnoisechisel input.fits --qthresh=0.4 --checkdetection

## Detect signal assuming input has 4 amplifier channels along first
## dimension and 1 along the second. Also set the regular tile size
## to 100 along both dimensions:
$ astnoisechisel --numchannels=4,1 --tilesize=100,100 input.fits

If NoiseChisel is to do processing (for example you don’t want to get help, or see the values to each input parameter), an input image should be provided with the recognized extensions (see Section 4.1.1.1 [Arguments], page 154). NoiseChisel shares a large set of common operations with other Gnuastro programs, mainly regarding input/output, general processing steps, and general operating modes. To help in a unified experience between all of Gnuastro’s programs, these operations have the same command-line options, see Section 4.1.2 [Common options], page 156, for a full list/description (they are not repeated here).

As in all Gnuastro programs, options can also be given to NoiseChisel in configuration files. For a thorough description on Gnuastro’s configuration file parsing, please see Section 4.2 [Configuration files], page 169. All of NoiseChisel’s options with a short description are also always available on the command-line with the --help option, see Section 4.3 [Getting help], page 172. To inspect the option values without actually running NoiseChisel, append your command with --printparams (or -P).
NoiseChisel’s input image may contain blank elements (see Section 6.1.3 [Blank pixels], page 262). Blank elements will be ignored in all steps of NoiseChisel. Hence if your dataset has bad pixels which should be masked with a mask image, please use Gnuastro’s Section 6.2 [Arithmetic], page 272, program (in particular its where operator) to convert those pixels to blank pixels before running NoiseChisel. Gnuastro’s Arithmetic program has bitwise operators helping you select specific kinds of bad-pixels when necessary.

A convolution kernel can also be optionally given. If a value (file name) is given to --kernel on the command-line or in a configuration file (see Section 4.2 [Configuration files], page 169), then that file will be used to convolve the image prior to thresholding. Otherwise a default kernel will be used. For a 2D image, the default kernel is a 2D Gaussian with a FWHM of 2 pixels truncated at 5 times the FWHM. This choice of the default kernel is discussed in Section 3.1.1 of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664). For a 3D cube, it is a Gaussian with FWHM of 1.5 pixels in the first two dimensions and 0.75 pixels in the third dimension. See Section 6.3.4 [Convolution kernel], page 324, for kernel related options. Passing none to --kernel will disable convolution. On the other hand, through the --convolved option, you may provide an already convolved image, see descriptions below for more.

NoiseChisel defines two tessellations over the input (see Section 4.8 [Tessellation], page 189). This enables it to deal with possible gradients in the input dataset and also significantly improve speed by processing each tile on different threads simultaneously. Tessellation related options are discussed in Section 4.1.2.2 [Processing options], page 159. In particular, NoiseChisel uses two tessellations (with everything between them identical except the tile sizes): a fine-grained one with smaller tiles (used in thresholding and Sky value estimations) and another with larger tiles which is used for pseudo-detections over non-detected regions of the image. The common Tessellation options described in Section 4.1.2.2 [Processing options], page 159, define all parameters of both tessellations. The large tile size for the latter tessellation is set through the --largetilesizes option. To inspect the tessellations on your input dataset, run NoiseChisel with --checktiles.

Usage TIP: Frequently use the options starting with --check. Since the noise properties differ between different datasets, you can often play with the parameters/options for a better result than the default parameters. You can start with --checkdetection for the main steps. For the full list of NoiseChisel’s checking options please run:

```
$ astnoisechisel --help | grep check
```

When working on 3D datacubes, the tessellation options need three values and updating them every time can be annoying/buggy. To simplify the job, NoiseChisel also installs a astnoisechisel-3d.conf configuration file (see Section 4.2 [Configuration files], page 169). You can use this for default values on datacubes. For example, if you installed Gnuastro with the prefix /usr/local (the default location, see Section 3.3.1.2 [Installation directory], page 139), you can benefit from this configuration file by running NoiseChisel like the example below.

```
$ astnoisechisel cube.fits
  --config=/usr/local/etc/astnoisechisel-3d.conf
```

To further simplify the process, you can define a shell alias in any startup file (for example ~/.bashrc, see Section 3.3.1.2 [Installation directory], page 139). Assuming that
you installed Gnuastro in /usr/local, you can add this line to the startup file (you may put it all in one line, it is broken into two lines here for fitting within page limits).

```
alias astnoisechisel-3d="astnoisechisel
    --config=/usr/local/etc/astnoisechisel-3d.conf"
```

Using this alias, you can call NoiseChisel with the name astnoisechisel-3d (instead of astnoisechisel). It will automatically load the 3D specific configuration file first, and then parse any other arguments, options or configuration files. You can change the default values in this 3D configuration file by calling them on the command-line as you do with astnoisechisel9. For example:

```
$ astnoisechisel-3d --numchannels=3,3,1 cube.fits
```

In the sections below, NoiseChisel’s options are classified into three general classes to help in easy navigation. Section 7.2.2.1 [NoiseChisel input], page 367, mainly discusses the options relating to input and those that are shared in both detection and segmentation. Options to configure the detection are described in Section 7.2.2.2 [Detection options], page 370, and Section 7.3.1.2 [Segmentation options], page 387, we discuss how you can fine-tune the segmentation of the detections. Finally, in Section 7.2.2.3 [NoiseChisel output], page 378, the format of NoiseChisel’s output is discussed. The order of options here follow the same logical order that the respective action takes place within NoiseChisel (note that the output of --help is sorted alphabetically).

Below, we’ll discuss NoiseChisel’s options, classified into two general classes, to help in easy navigation. Section 7.2.2.1 [NoiseChisel input], page 367, mainly discusses the basic options relating to inputs and prior to the detection process detection. Afterwards, Section 7.2.2.2 [Detection options], page 370, fully describes every configuration parameter (option) related to detection and how they affect the final result. The order of options in this section follow the logical order within NoiseChisel. On first reading (while you are still new to NoiseChisel), it is therefore strongly recommended to read the options in the given order below. The output of --printparams (or -P) also has this order. However, the output of --help is sorted alphabetically. Finally, in Section 7.2.2.3 [NoiseChisel output], page 378, the format of NoiseChisel’s output is discussed.

### 7.2.2.1 NoiseChisel input

The options here can be used to configure the inputs and output of NoiseChisel, along with some general processing options. Recall that you can always see the full list of Gnuastro’s options with the --help (see Section 4.3 [Getting help], page 172), or --printparams (or -P) to see their values (see Section 4.1.2.3 [Operating mode options], page 162).

```
-k FITS
    --kernel=FITS
```

File name of kernel to smooth the image before applying the threshold, see Section 6.3.4 [Convolution kernel], page 324. If no convolution is needed, give this option a value of none.

The first step of NoiseChisel is to convolve/smooth the image and use the convolved image in multiple steps including the finding and applying of the

---

9 Recall that for single-invocation options, the last command-line invocation takes precedence over all previous invocations (including those in the 3D configuration file). See the description of --config in Section 4.1.2.3 [Operating mode options], page 162.
quantile threshold (see `--qthresh`). The `--kernel` option is not mandatory. If not called, for a 2D, image a 2D Gaussian profile with a FWHM of 2 pixels truncated at 5 times the FWHM is used. This choice of the default kernel is discussed in Section 3.1.1 of Akhlaghi and Ichikawa [2015].

For a 3D cube, when no file name is given to `--kernel`, a Gaussian with FWHM of 1.5 pixels in the first two dimensions and 0.75 pixels in the third dimension will be used. The reason for this particular configuration is that commonly in astronomical applications, 3D datasets don’t have the same nature in all three dimensions, commonly the first two dimensions are spatial (RA and Dec) while the third is spectral (for example wavelength). The samplings are also different, in the default case, the spatial sampling is assumed to be larger than the spectral sampling, hence a wider FWHM in the spatial directions, see Section 6.3.2.7 [Sampling theorem], page 318.

You can use MakeProfiles to build a kernel with any of its recognized profile types and parameters. For more details, please see Section 8.1.4.3 [MakeProfiles output dataset], page 457. For example, the command below will make a Moffat kernel (with $\beta = 2.8$) with FWHM of 2 pixels truncated at 10 times the FWHM.

```
$ astmkprof --oversample=1 --kernel=moffat,2,2.8,10
```

Since convolution can be the slowest step of NoiseChisel, for large datasets, you can convolve the image once with Gnuastro’s Convolve (see Section 6.3 [Convolve], page 306), and use the `--convolved` option to feed it directly to NoiseChisel. This can help getting faster results when you are playing/testing the higher-level options.

```
--khdu=STR
```

HDU containing the kernel in the file given to the `--kernel` option.

```
--convolved=FITS
```

Use this file as the convolved image and don’t do convolution (ignore `--kernel`). NoiseChisel will just check the size of the given dataset is the same as the input’s size. If a wrong image (with the same size) is given to this option, the results (errors, bugs, etc) are unpredictable. So please use this option with care and in a highly controlled environment, for example in the scenario discussed below.

In almost all situations, as the input gets larger, the single most CPU (and time) consuming step in NoiseChisel (and other programs that need a convolved image) is convolution. Therefore minimizing the number of convolutions can save a significant amount of time in some scenarios. One such scenario is when you want to segment NoiseChisel’s detections using the same kernel (with Section 7.3 [Segment], page 381, which also supports this `--convolved` option). This scenario would require two convolutions of the same dataset: once by NoiseChisel and once by Segment. Using this option in both programs, only one convolution (prior to running NoiseChisel) is enough.

Another common scenario where this option can be convenient is when you are testing NoiseChisel (or Segment) for the best parameters. You have to run NoiseChisel multiple times and see the effect of each change. However, once you are happy with the kernel, re-convolving the input on every change of higher-level parameters will greatly hinder, or discourage, further testing. With this
option, you can convolve the input image with your chosen kernel once before running NoiseChisell, then feed it to NoiseChisell on each test run and thus save valuable time for better/more tests.

To build your desired convolution kernel, you can use Section 8.1 [MakeProfiles], page 440. To convolve the image with a given kernel you can use Section 6.3 [Convolve], page 306. Spatial domain convolution is mandatory: in the frequency domain, blank pixels (if present) will cover the whole image and gradients will appear on the edges, see Section 6.3.3 [Spatial vs. Frequency domain], page 323.

Below you can see an example of the second scenario: you want to see how variation of the growth level (through the --detgrowquant option) will affect the final result. Recall that you can ignore all the extra spaces, new lines, and backslash’s ('\') if you are typing in the terminal. In a shell script, remove the $ signs at the start of the lines.

## Make the kernel to convolve with.
$ astmkprof --oversample=1 --kernel=gaussian,2,5

## Convolve the input with the given kernel.
$ astconvolve input.fits --kernel=kernel.fits --domain=spatial --output=convolved.fits

## Run NoiseChisell with seven growth quantile values.
$ for g in 60 65 70 75 80 85 90; do
    astnoisechisell input.fits --convolved=convolved.fits --detgrowquant=0.$g --output=$g.fits; done

--chdu=STR
The HDU/extension containing the convolved image in the file given to --convolved.

-w FITS
--widekernel=FITS

File name of a wider kernel to use in estimating the difference of the mode and median in a tile (this difference is used to identify the significance of signal in that tile, see Section 7.1.4.3 [Quantifying signal in a tile], page 347). As displayed in Figure 4 of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664), a wider kernel will help in identifying the skewness caused by data in noise. The image that is convolved with this kernel is only used for this purpose. Once the mode is found to be sufficiently close to the median, the quantile threshold is found on the image convolved with the sharper kernel (--kernel), see --qthresh).

Since convolution will significantly slow down the processing, this feature is optional. When it isn’t given, the image that is convolved with --kernel will be used to identify good tiles and apply the quantile threshold. This option is mainly useful in conditions where you have a very large, extended, diffuse signal that is still present in the usable tiles when using --kernel. See Section 2.3
[Detecting large extended targets], page 76, for a practical demonstration on how to inspect the tiles used in identifying the quantile threshold.

\[ \text{--whdu=STR} \]
HDU containing the kernel file given to the \text{--widekernel} option.

\[ \text{-L INT[,INT]} \]
\[ \text{--largetilesizesize=INT[,INT]} \]
The size of each tile for the tessellation with the larger tile sizes. Except for the tile size, all the other parameters for this tessellation are taken from the common options described in Section 4.1.2.2 [Processing options], page 159. The format is identical to that of the \text{--tilesizesize} option that is discussed in that section.

### 7.2.2.2 Detection options

Detection is the process of separating the pixels in the image into two groups: 1) Signal, and 2) Noise. Through the parameters below, you can customize the detection process in NoiseChisel. Recall that you can always see the full list of NoiseChisel’s options with the \text{--help} (see Section 4.3 [Getting help], page 172), or \text{--printparams} (or \text{-P}) to see their values (see Section 4.1.2.3 [Operating mode options], page 162).

\[ \text{-Q FLT} \]
\[ \text{--meanmedqdiff=FLT} \]
The maximum acceptable distance between the quantiles of the mean and median in each tile, see Section 7.1.4.3 [Quantifying signal in a tile], page 347. The quantile threshold estimates are measured on tiles where the quantiles of their mean and median are less distant than the value given to this option. For example \text{--meanmedqdiff=0.01} means that only tiles where the mean’s quantile is between 0.49 and 0.51 (recall that the median’s quantile is 0.5) will be used.

\[ \text{--outliersclip=FLT,FLT} \]
\[ \sigma \]-clipping parameters for the outlier rejection of the quantile threshold. The format of the given values is similar to \text{--sigmaclip} below. In NoiseChisel, outlier rejection on tiles is used when identifying the quantile thresholds (\text{--qthresh}, \text{--noerodequant}, and \text{detgrowquant}). Outlier rejection is useful when the dataset contains a large and diffuse (almost flat within each tile) signal. The flatness of the profile will cause it to successfully pass the mean-median quantile difference test, so we’ll need to use the distribution of successful tiles for removing these false positives. For more, see the latter half of Section 7.1.4.3 [Quantifying signal in a tile], page 347.

\[ \text{--outlierssigma=FLT} \]
Multiple of sigma to define an outlier. If this option is given a value of zero, no outlier rejection will take place. For more see \text{--outliersclip} and the latter half of Section 7.1.4.3 [Quantifying signal in a tile], page 347.

\[ \text{-t FLT} \]
\[ \text{--qthresh=FLT} \]
The quantile threshold to apply to the convolved image. The detection process begins with applying a quantile threshold to each of the tiles in the small tessellation. The quantile is only calculated for tiles that don’t have any significant
signal within them, see Section 7.1.4.3 [Quantifying signal in a tile], page 347. Interpolation is then used to give a value to the unsuccessful tiles and it is finally smoothed.

The quantile value is a floating point value between 0 and 1. Assume that we have sorted the $N$ data elements of a distribution (the pixels in each mesh on the convolved image). The quantile ($q$) of this distribution is the value of the element with an index of (the nearest integer to) $q \times N$ in the sorted data set. After thresholding is complete, we will have a binary (two valued) image. The pixels above the threshold are known as foreground pixels (have a value of 1) while those which lie below the threshold are known as background (have a value of 0).

```
--smoothwidth=INT
Width of flat kernel used to smooth the interpolated quantile thresholds, see --qthresh for more.
```

```
--checkqthresh
Check the quantile threshold values on the mesh grid. A multi-extension FITS file, suffixed with _qthresh.fits will be created showing each step of how the final quantile threshold is found. With this option, NoiseChisel will abort as soon as quantile estimation has been completed, allowing you to inspect the steps leading to the final quantile threshold, this can be disabled with --continueaftercheck. By default the output will have the same pixel size as the input, but with the --oneelempertile option, only one pixel will be used for each tile (see Section 4.1.2.2 [Processing options], page 159).
```

The key things to remember are:

- The measurements to find the thresholds are done on tiles that cover the whole image in a tessellation. Recall that you can set the size of tiles with --tilesize and check them with --checktiles. Therefore except for the first and last extensions, the rest only show tiles.

- NoiseChisel ultimately has three thresholds: the quantile threshold (that you set with --qthresh), the no-erode quantile (set with --noerodequant) and the growth quantile (set with --detgrowquant). Therefore for each step, we have three extensions.

The output file will have the following extensions. Below, the extensions are put in the same order as you see in the file, with their name.

```
CONVOLVED
This is the input image after convolution with the kernel (which is a FWHM=2 Gaussian by default, but you can change with --kernel). Recall that the thresholds are defined on the convolved image.
```

```
QTHRESH_ERODE
QTHRESH_NOERODE
QTHRESH_EXPAND
In these three extensions, the tiles that have a quantile-of-mean more/less than 0.5 (quantile of median) $\pm d$ are set to NaN ($d$ is
the value given to \texttt{--meanmedqdiff}, see Section 7.1.4.3 [Quantifying signal in a tile], page 347). Therefore the non-NaN tiles that you see here are the tiles where there is no significant skewness (changing signal) within that tile. The only differing thing between the three extensions is the values of the non-NaN tiles. These values will be used to construct the final threshold map over the whole image.

\begin{verbatim}
VALUE1_NO_OUTLIER
VALUE2_NO_OUTLIER
VALUE3_NO_OUTLIER
\end{verbatim}

All outlier tiles have been masked. The reason for removing outliers is that the quantile-of-mean is only sensitive to signal that varies on a scale that is smaller than the tile size. Therefore the extended wings of large galaxies or bright stars (which vary on scales much larger than the tile size) will pass that test. As described in Section 7.1.4.3 [Quantifying signal in a tile], page 347, using the values given to \texttt{--outliersclip} and \texttt{--outliersigma}, the outlier tiles are removed in this step.

\begin{verbatim}
THRESH1_INTERP
THRESH2_INTERP
THRESH3_INTERP
\end{verbatim}

Using the successful values that remain after the previous step, give values to all (interpolate) the tiles in the image. The interpolation is done using the nearest-neighbor method: for each tile, the N nearest neighbors are found and the median of their values is used to fill it. You can set the value of N through the \texttt{--interpnumngb} option.

\begin{verbatim}
THRESH1_SMOOTH
THRESH2_SMOOTH
THRESH3_SMOOTH
\end{verbatim}

Smooth the interpolated image to remove the strong differences between touching tiles. Because we used the median value of the N nearest neighbors in the previous step, there can be strong discontinuities on the edges of tiles (which can directly show in the image after applying the threshold). The scale of the smoothing (number of nearby tiles to smooth with) is set with the \texttt{--smoothwidth} option.

\begin{verbatim}
QTHRESH-APPLIED
\end{verbatim}

The pixels in this image can only have three values:

0 These pixels had a value below the quantile threshold.

1 These pixels had a value above the quantile threshold, but below the threshold for no erosion. Therefore in the next step, NoiseChisel will erode (set them to 0) these pixels if they are touching a 0-valued pixel.
These pixels had a value above the no-erosion threshold. So NoiseChisel will not erode these pixels, it will only apply Opening to them afterwards. Recall that this was done to avoid losing sharp point-sources (like stars in space-based imaging).

--blankasforeground
In the erosion and opening steps below, treat blank elements as foreground (regions above the threshold). By default, blank elements in the dataset are considered to be background, so if a foreground pixel is touching it, it will be eroded. This option is irrelevant if the datasets contains no blank elements. When there are many blank elements in the dataset, treating them as foreground will systematically erode their regions less, therefore systematically creating more false positives. So use this option (when blank values are present) with care.

-e INT
--erode=INT
The number of erosions to apply to the binary thresholded image. Erosion is simply the process of flipping (from 1 to 0) any of the foreground pixels that neighbor a background pixel. In a 2D image, there are two kinds of neighbors, 4-connected and 8-connected neighbors. In a 3D dataset, there are three: 6-connected, 18-connected, and 26-connected. You can specify which class of neighbors should be used for erosion with the --erodengb option, see below.
Erosion has the effect of shrinking the foreground pixels. To put it another way, it expands the holes. This is a founding principle in NoiseChisel: it exploits the fact that with very low thresholds, the holes in the very low surface brightness regions of an image will be smaller than regions that have no signal. Therefore by expanding those holes, we are able to separate the regions harboring signal.

--erodengb=INT
The type of neighborhood (structuring element) used in erosion, see --erode for an explanation on erosion. If the input is a 2D image, only two integer values are acceptable: 4 or 8. For a 3D input datacube, the acceptable values are: 6, 18 and 26.
In 2D 4-connectivity, the neighbors of a pixel are defined as the four pixels on the top, bottom, right and left of a pixel that share an edge with it. The 8-connected neighbors on the other hand include the 4-connected neighbors along with the other 4 pixels that share a corner with this pixel. See Figure 6 (a) and (b) in Akhlaghi and Ichikawa (2015) for a demonstration. A similar argument applies to 3D datacubes.

--noerodequant
Pure erosion is going to carve off sharp and small objects completely out of the detected regions. This option can be used to avoid missing such sharp and small objects (which have significant pixels, but not over a large area). All pixels with a value larger than the significance level specified by this option will not be eroded during the erosion step above. However, they will undergo the erosion and dilation of the opening step below.
Like the --qthresh option, the significance level is determined using the quantile (a value between 0 and 1). Just as a reminder, in the normal distribution, $1\sigma$, $1.5\sigma$, and $2\sigma$ are approximately on the 0.84, 0.93, and 0.98 quantiles.

-p INT
--opening=INT

Depth of opening to be applied to the eroded binary image. Opening is a composite operation. When opening a binary image with a depth of $n\sigma$, $n$ erosions (explained in --erode) are followed by $n$ dilations. Simply put, dilation is the inverse of erosion. When dilating an image any background pixel is flipped (from 0 to 1) to become a foreground pixel. Dilation has the effect of fattening the foreground. Note that in NoiseChisel, the erosion which is part of opening is independent of the initial erosion that is done on the thresholded image (explained in --erode). The structuring element for the opening can be specified with the --openingngb option. Opening has the effect of removing the thin foreground connections (mostly noise) between separate foreground 'islands' (detections) thereby completely isolating them. Once opening is complete, we have initial detections.

--openingngb=INT

The structuring element used for opening, see --erodengb for more information about a structuring element.

--skyfracnoblank

Ignore blank pixels when estimating the fraction of undetected pixels for Sky estimation. NoiseChisel only measures the Sky over the tiles that have a sufficiently large fraction of undetected pixels (value given to --minskyfrac). By default this fraction is found by dividing number of undetected pixels in a tile by the tile's area. But this default behavior ignores the possibility of blank pixels. In situations that blank/masked pixels are scattered across the image and if they are large enough, all the tiles can fail the --minskyfrac test, thus not allowing NoiseChisel to proceed. With this option, such scenarios can be fixed: the denominator of the fraction will be the number of non-blank elements in the tile, not the total tile area.

-B FLT
--minskyfrac=FLT

Minimum fraction (value between 0 and 1) of Sky (undetected) areas in a tile. Only tiles with a fraction of undetected pixels (Sky) larger than this value will be used to estimate the Sky value. NoiseChisel uses this option value twice to estimate the Sky value: after initial detections and in the end when false detections have been removed.

Because of the PSF and their intrinsic amorphous properties, astronomical objects (except cosmic rays) never have a clear cutoff and commonly sink into the noise very slowly. Even below the very low thresholds used by NoiseChisel. So when a large fraction of the area of one mesh is covered by detections, it is very plausible that their faint wings are present in the undetected regions (hence causing a bias in any measurement). To get an accurate measurement of the above parameters over the tessellation, tiles that harbor too many detected
regions should be excluded. The used tiles are visible in the respective --check option of the given step.

--checkdetsky
Check the initial approximation of the sky value and its standard deviation in a FITS file ending with _detsky.fits. With this option, NoiseChisel will abort as soon as the sky value used for defining pseudo-detections is complete. This allows you to inspect the steps leading to the final quantile threshold, this behavior can be disabled with --continueaftercheck. By default the output will have the same pixel size as the input, but with the --oneelementperTile option, only one pixel will be used for each tile (see Section 4.1.2.2 [Processing options], page 159).

-s FLT,FLT
--sigmaclip=FLT,FLT
The $\sigma$-clipping parameters for measuring the initial and final Sky values from the undetected pixels, see Section 7.1.3 [Sigma clipping], page 343. This option takes two values which are separated by a comma (,). Each value can either be written as a single number or as a fraction of two numbers (for example 3,1/10). The first value to this option is the multiple of $\sigma$ that will be clipped ($\alpha$ in that section). The second value is the exit criteria. If it is less than 1, then it is interpreted as tolerance and if it is larger than one it is assumed to be the fixed number of iterations. Hence, in the latter case the value must be an integer.

-R FLT
--dthresh=FLT
The detection threshold: a multiple of the initial Sky standard deviation added with the initial Sky approximation (which you can inspect with --checkdetsky). This flux threshold is applied to the initially undetected regions on the unconvolved image. The background pixels that are completely engulfed in a 4-connected foreground region are converted to background (holes are filled) and one opening (depth of 1) is applied over both the initially detected and undetected regions. The Signal to noise ratio of the resulting ‘pseudo-detections’ are used to identify true vs. false detections. See Section 3.1.5 and Figure 7 in Akhlaghi and Ichikawa (2015) for a very complete explanation.

--dopening=INT
The number of openings to do after applying --dthresh.

--dopeningngb=INT
The connectivity used in the opening of --dopening. In a 2D image this must be either 4 or 8. The stronger the connectivity, the more smaller regions will be discarded.

--holengb=INT
The connectivity (defined by the number of neighbors) to fill holes after applying --dthresh (above) to find pseudo-detections. For example in a 2D image it must be 4 (the neighbors that are most strongly connected) or 8 (all neighbors).
The stronger the connectivity, the stronger the hole will be enclosed. So setting a value of 8 in a 2D image means that the walls of the hole are 4-connected. If standard (near Sky level) values are given to \texttt{-dthresh}, setting \texttt{-holengb=4}, might fill the complete dataset and thus not create enough pseudo-detections.

\texttt{--pseudoconcomp=INT}

The connectivity (defined by the number of neighbors) to find individual pseudo-detections. If it is a weaker connectivity (4 in a 2D image), then pseudo-detections that are connected on the corners will be treated as separate.

\texttt{-m INT}

\texttt{--smminarea=INT}

The minimum area to calculate the Signal to noise ratio on the pseudo-detections of both the initially detected and undetected regions. When the area in a pseudo-detection is too small, the Signal to noise ratio measurements will not be accurate and their distribution will be heavily skewed to the positive. So it is best to ignore any pseudo-detection that is smaller than this area. Use \texttt{--detsnhistnbins} to check if this value is reasonable or not.

\texttt{--checksn}

Save the S/N values of the pseudo-detections (and possibly grown detections if \texttt{--cleangrounddet} is called) into separate tables. If \texttt{--tableformat} is a FITS table, each table will be written into a separate extension of one file suffixed with \texttt{_detsn.fits}. If it is plain text, a separate file will be made for each table (ending in \texttt{_detsn_sky.txt}, \texttt{_detsn_det.txt} and \texttt{_detsn_grown.txt}). For more on \texttt{--tableformat} see Section 4.1.2.1 [Input/Output options], page 157. You can use these to inspect the S/N values and their distribution (in combination with the \texttt{--checkdetection} option to see where the pseudo-detections are). You can use Gnuastro’s Section 7.1 [Statistics], page 337, to make a histogram of the distribution or any other analysis you would like for better understanding of the distribution (for example through a histogram).

\texttt{--minnumfalse=INT}

The minimum number of ‘pseudo-detections’ over the undetected regions to identify a Signal-to-Noise ratio threshold. The Signal to noise ratio (S/N) of false pseudo-detections in each tile is found using the quantile of the S/N distribution of the pseudo-detections over the undetected pixels in each mesh. If the number of S/N measurements is not large enough, the quantile will not be accurate (can have large scatter). For example if you set \texttt{--snquant=0.99} (or the top 1 percent), then it is best to have at least 100 S/N measurements.

\texttt{-c FLT}

\texttt{--snquant=FLT}

The quantile of the Signal to noise ratio distribution of the pseudo-detections in each mesh to use for filling the large mesh grid. Note that this is only calculated for the large mesh grids that satisfy the minimum fraction of undetected pixels (value of \texttt{--minbfrac}) and minimum number of pseudo-detections (value of \texttt{--minnumfalse}).
---snthresh=FLT
Manually set the signal-to-noise ratio of true pseudo-detections. With this option, NoiseChisel will not attempt to find pseudo-detections over the noisy regions of the dataset, but will directly go onto applying the manually input value.

This option is useful in crowded images where there is no blank sky to find the sky pseudo-detections. You can get this value on a similarly reduced dataset (from another region of the Sky with more undetected regions spaces).

-d FLT
---detgrowquant=FLT
Quantile limit to “grow” the final detections. As discussed in the previous options, after applying the initial quantile threshold, layers of pixels are carved off the objects to identify true signal. With this step you can return those low surface brightness layers that were carved off back to the detections. To disable growth, set the value of this option to 1.

The process is as follows: after the true detections are found, all the non-detected pixels above this quantile will be put in a list and used to “grow” the true detections (seeds of the growth). Like all quantile thresholds, this threshold is defined and applied to the convolved dataset. Afterwards, the dataset is dilated once (with minimum connectivity) to connect very thin regions on the boundary: imagine building a dam at the point rivers spill into an open sea/ocean. Finally, all holes are filled. In the geography metaphor, holes can be seen as the closed (by the dams) rivers and lakes, so this process is like turning the water in all such rivers and lakes into soil. See ---detgrowmaxholesize for configuring the hole filling.

Note that since the growth occurs on all neighbors of a data element, the quantile for 3D detection must be must larger than that of 2D detection. Recall that in 2D each element has 8 neighbors while in 3D there are 27 neighbors.

---detgrowmaxholesize=INT
The maximum hole size to fill during the final expansion of the true detections as described in ---detgrowquant. This is necessary when the input contains many smaller objects and can be used to avoid marking blank sky regions as detections.

For example multiple galaxies can be positioned such that they surround an empty region of sky. If all the holes are filled, the Sky region in between them will be taken as a detection which is not desired. To avoid such cases, the integer given to this option must be smaller than the hole between such objects. However, we should caution that unless the “hole” is very large, the combined faint wings of the galaxies might actually be present in between them, so be very careful in not filling such holes.

On the other hand, if you have a very large (and extended) galaxy, the diffuse wings of the galaxy may create very large holes over the detections. In such cases, a large enough value to this option will cause all such holes to be detected as part of the large galaxy and thus help in detecting it to extremely low surface brightness limits. Therefore, especially when large and extended objects are
present in the image, it is recommended to give this option (very) large values. For one real-world example, see Section 2.3 [Detecting large extended targets], page 76.

--cleangrowndet
After dilation, if the signal-to-noise ratio of a detection is less than the derived pseudo-detection S/N limit, that detection will be discarded. In an ideal/clean noise, a true detection’s S/N should be larger than its constituent pseudo-detections because its area is larger and it also covers more signal. However, on a false detections (especially at lower --snquant values), the increase in size can cause a decrease in S/N below that threshold. This will improve purity and not change completeness (a true detection will not be discarded). Because a true detection has flux in its vicinity and dilation will catch more of that flux and increase the S/N. So on a true detection, the final S/N cannot be less than pseudo-detections. However, in many real images bad processing creates artifacts that cannot be accurately removed by the Sky subtraction. In such cases, this option will decrease the completeness (will artificially discard true detections). So this feature is not default and should to be explicitly called when you know the noise is clean.

--checkdetection
Every step of the detection process will be added as an extension to a file with the suffix _det.fits. Going through each would just be a repeat of the explanations above and also of those in Akhlaghi and Ichikawa (2015). The extension label should be sufficient to recognize which step you are observing. Viewing all the steps can be the best guide in choosing the best set of parameters. With this option, NoiseChisel will abort as soon as a snapshot of all the detection process is saved. This behavior can be disabled with --continueaftercheck.

--checksky
Check the derivation of the final sky and its standard deviation values on the mesh grid. With this option, NoiseChisel will abort as soon as the sky value is estimated over the image (on each tile). This behavior can be disabled with --continueaftercheck. By default the output will have the same pixel size as the input, but with the --oneelempertile option, only one pixel will be used for each tile (see Section 4.1.2.2 [Processing options], page 159).

7.2.2.3 NoiseChisel output
NoiseChisel’s output is a multi-extension FITS file. The main extension/dataset is a (binary) detection map. It has the same size as the input but with only two possible values for all pixels: 0 (for pixels identified as noise) and 1 (for those identified as signal/detections). The detection map is followed by a Sky and Sky standard deviation dataset (which are calculated from the binary image). By default (when --rawoutput isn’t called), NoiseChisel will also subtract the Sky value from the input and save the sky-subtracted input as the first extension in the output with data. The zero-th extension (that contains no data), contains NoiseChisel’s configuration as FITS keywords, see Section 4.10 [Output FITS files], page 191.
The name of the output file can be set by giving a value to `--output` (this is a common option between all programs and is therefore discussed in Section 4.1.2.1 [Input/Output options], page 157). If `--output` isn’t used, the input name will be suffixed with `_detected.fits` and used as output, see Section 4.9 [Automatic output], page 191. If any of the options starting with `--check*` are given, NoiseChisel won’t complete and will abort as soon as the respective check images are created. For more information on the different check images, see the description for the `--check*` options in Section 7.2.2.2 [Detection options], page 370, (this can be disabled with `--continueaftercheck`).

The last two extensions of the output are the Sky and its Standard deviation, see Section 7.1.4 [Sky value], page 344, for a complete explanation. They are calculated on the tile grid that you defined for NoiseChisel. By default these datasets will have the same size as the input, but with all the pixels in one tile given one value. To be more space-efficient (keep only one pixel per tile), you can use the `--oneelempertile` option, see Section 4.8 [Tessellation], page 189.

To inspect any of NoiseChisel’s output files, assuming you use SAO DS9, you can configure your Graphic User Interface (GUI) to open NoiseChisel’s output as a multi-extension data cube. This will allow you to flip through the different extensions and visually inspect the results. This process has been described for the GNOME GUI (most common GUI in GNU/Linux operating systems) in Section 10.4 [Viewing FITS file contents with DS9 or TOPCAT], page 494.

NoiseChisel’s output configuration options are described in detail below.

`--continueaftercheck`  
Continue NoiseChisel after any of the options starting with `--check` (see Section 7.2.2.2 [Detection options], page 370. NoiseChisel involves many steps and as a result, there are many checks, allowing you to inspect the status of the processing. The results of each step affect the next steps of processing. Therefore, when you want to check the status of the processing at one step, the time spent to complete NoiseChisel is just wasted/distracting time.

To encourage easier experimentation with the option values, when you use any of the NoiseChisel options that start with `--check`, NoiseChisel will abort once its desired extensions have been written. With `--continueaftercheck` option, you can disable this behavior and ask NoiseChisel to continue with the rest of the processing, even after the requested check files are complete.

`--ignoreblankintiles`  
Don’t set the input’s blank pixels to blank in the tiled outputs (for example Sky and Sky standard deviation extensions of the output). This is only applicable when the tiled output has the same size as the input, in other words, when `--oneelempertile` isn’t called.

By default, blank values in the input (commonly on the edges which are outside the survey/field area) will be set to blank in the tiled outputs also. But in other scenarios this default behavior is not desired: for example if you have masked something in the input, but want the tiled output under that also.

`-l`  
`--label` Run a connected-components algorithm on the finally detected pixels to identify which pixels are connected to which. By default the main output is a binary
dataset with only two values: 0 (for noise) and 1 (for signal/detections). See Section 7.2.2.3 [NoiseChisel output], page 378, for more.

The purpose of NoiseChisel is to detect targets that are extended and diffuse, with outer parts that sink into the noise very gradually (galaxies and stars for example). Since NoiseChisel digs down to extremely low surface brightness values, many such targets will commonly be detected together as a single large body of connected pixels.

To properly separate connected objects, sophisticated segmentation methods are commonly necessary on NoiseChisel’s output. Gnuastro has the dedicated Section 7.3 [Segment], page 381, program for this job. Since input images are commonly large and can take a significant volume, the extra volume necessary to store the labels of the connected components in the detection map (which will be created with this --label option, in 32-bit signed integer type) can thus be a major waste of space. Since the default output is just a binary dataset, an 8-bit unsigned dataset is enough.

The binary output will also encourage users to segment the result separately prior to doing higher-level analysis. As an alternative to --label, if you have the binary detection image, you can use the connected-components operator in Gnuastro’s Arithmetic program to identify regions that are connected with each other. For example with this command (assuming NoiseChisel’s output is called nc.fits):

$ astarithmetic nc.fits 2 connected-components -hDETECTIONS

--rawoutput

Don’t include the Sky-subtracted input image as the first extension of the output. By default, the Sky-subtracted input is put in the first extension of the output. The next extensions are NoiseChisel’s main outputs described above.

The extra Sky-subtracted input can be convenient in checking NoiseChisel’s output and comparing the detection map with the input: visually see if everything you expected is detected (reasonable completeness) and that you don’t have too many false detections (reasonable purity). This visual inspection is simplified if you use SAO DS9 to view NoiseChisel’s output as a multi-extension data-cube, see Section 10.4 [Viewing FITS file contents with DS9 or TOPCAT], page 494.

When you are satisfied with your NoiseChisel configuration (therefore you don’t need to check on every run), or you want to archive/transfer the outputs, or the datasets become large, or you are running NoiseChisel as part of a pipeline, this Sky-subtracted input image can be a significant burden (take up a large volume). The fact that the input is also noisy, makes it hard to compress it efficiently.

In such cases, this --rawoutput can be used to avoid the extra sky-subtracted input in the output. It is always possible to easily produce the Sky-subtracted dataset from the input (assuming it is in extension 1 of in.fits) and the SKY extension of NoiseChisel’s output (let’s call it nc.fits) with a command like below (assuming NoiseChisel wasn’t run with --oneelementtile, see Section 4.8 [Tessellation], page 189):

$ astarithmetic nc.fits 2 connected-components --rawoutput
$ astarithmetic in.fits nc.fits -h1 -hSKY

Save space: with the --rawoutput and --oneelempertile, NoiseChisel’s output will only be one binary detection map and two much smaller arrays with one value per tile. Since none of these have noise they can be compressed very effectively (without any loss of data) with exceptionally high compression ratios. This makes it easy to archive, or transfer, NoiseChisel’s output even on huge datasets. To compress it with the most efficient method (take up less volume), run the following command:

$ gzip --best noisechisel_output.fits

The resulting .fits.gz file can then be fed into any of Gnuastro’s programs directly, or viewed in viewers like SAO DS9, without having to decompress it separately (they will just take a little longer, because they have to internally decompress it before starting). See Section 2.2.12 [NoiseChisel optimization for storage], page 51, for an example on a real dataset.

### 7.3 Segment

Once signal is separated from noise (for example with Section 7.2 [NoiseChisel], page 362), you have a binary dataset: each pixel is either signal (1) or noise (0). Signal (for example every galaxy in your image) has been “detected”, but all detections have a label of 1. Therefore while we know which pixels contain signal, we still can’t find out how many galaxies they contain or which detected pixels correspond to which galaxy. At the lowest (most generic) level, detection is a kind of segmentation (segmenting the whole dataset into signal and noise, see Section 7.2 [NoiseChisel], page 362). Here, we’ll define segmentation only on signal: to separate sub-structure within the detections.

If the targets are clearly separated, or their detected regions aren’t touching, a simple connected components algorithm (very basic segmentation) is enough to separate the regions that are touching/connected. This is such a basic and simple form of segmentation that Gnuastro’s Arithmetic program has an operator for it: see connected-components in Section 6.2.3 [Arithmetic operators], page 276. Assuming the binary dataset is called binary.fits, you can use it with a command like this:

$ astarithmetic binary.fits 2 connected-components

You can even do a very basic detection (a threshold, say at value 100) and segmentation in Arithmetic with a single command like below:

$ astarithmetic in.fits 100 gt 2 connected-components

However, in most astronomical situations our targets are not nicely separated or have a sharp boundary/edge (for a threshold to suffice): they touch (for example merging galaxies), or are simply in the same line-of-sight (which is much more common). This causes their images to overlap.

In particular, when you do your detection with NoiseChisel, you will detect signal to very low surface brightness limits: deep into the faint wings of galaxies or bright stars (which can extend very far and irregularly from their center). Therefore, it often happens that several galaxies are detected as one large detection. Since they are touching, a simple connected components

---

10 https://en.wikipedia.org/wiki/Connected-component_labeling
components algorithm will not suffice. It is therefore necessary to do a more sophisticated segmentation and break up the detected pixels (even those that are touching) into multiple target objects as accurately as possible.

Segment will use a detection map and its corresponding dataset to find sub-structure over the detected areas and use them for its segmentation. Until Gnuastro version 0.6 (released in 2018), Segment was part of Section 7.2 [NoiseChisel], page 362. Therefore, similar to NoiseChisel, the best place to start reading about Segment and understanding what it does (with many illustrative figures) is Section 3.2 of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664), and continue with Akhlaghi [2019] (https://arxiv.org/abs/1909.11230).

As a summary, Segment first finds true clumps over the detections. Clumps are associated with local maxima/minima\(^\text{11}\) and extend over the neighboring pixels until they reach a local minimum/maximum (river/watershed). By default, Segment will use the distribution of clump signal-to-noise ratios over the undetected regions as reference to find “true” clumps over the detections. Using the undetected regions can be disabled by directly giving a signal-to-noise ratio to --clumpsnthresh.

The true clumps are then grown to a certain threshold over the detections. Based on the strength of the connections (rivers/watersheds) between the grown clumps, they are considered parts of one object or as separate objects. See Section 3.2 of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664) for more. Segment’s main output are thus two labeled datasets: 1) clumps, and 2) objects. See Section 7.3.1.3 [Segment output], page 390, for more.

To start learning about Segment, especially in relation to detection (Section 7.2 [NoiseChisel], page 362) and measurement (Section 7.4 [MakeCatalog], page 392), the recommended references are Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664), Akhlaghi [2016] (https://arxiv.org/abs/1611.06387) and Akhlaghi [2019] (https://arxiv.org/abs/1909.11230). If you have used Segment within your research, please run it with --cite to list the papers you should cite and how to acknowledge its funding sources.

Those papers cannot be updated any more but the software will evolve. For example Segment became a separate program (from NoiseChisel) in 2018 (after those papers were published). Therefore this book is the definitive reference. Finally, in Section 7.3.1 [Invoking Segment], page 382, we’ll discuss Segment’s inputs, outputs and configuration options.

### 7.3.1 Invoking Segment

Segment will identify substructure within the detected regions of an input image. Segment’s output labels can be directly used for measurements (for example with Section 7.4 [MakeCatalog], page 392). The executable name is astsegment with the following general template

$ astsegment [OPTION ...] InputImage.fits

One line examples:

```
## Segment NoiseChisel's detected regions.
```

\(^\text{11}\) By default the maximum is used as the first clump pixel, to define clumps based on local minima, use the --minima option.
Chapter 7: Data analysis

$ astsegment default-noisechisel-output.fits

## Use a hand-input S/N value for keeping true clumps
## (avoid finding the S/N using the undetected regions).
$ astsegment nc-out.fits --clumpsnthresh=10

## Inspect all the segmentation steps after changing a parameter.
$ astsegment input.fits --snquant=0.9 --checksegmentation

## Use the fixed value of 0.01 for the input’s Sky standard deviation
## (in the units of the input), and assume all the pixels are a
detection (for example a large structure extending over the whole
## image), and only keep clumps with S/N>10 as true clumps.
$ astsegment in.fits --std=0.01 --detection=all --clumpsnthresh=10

If Segment is to do processing (for example you don’t want to get help, or see the values of
each option), at least one input dataset is necessary along with detection and error information,
either as separate datasets (per-pixel) or fixed values, see Section 7.3.1.1 [Segment input], page 383. Segment shares a large set of common operations with other Gnuastro programs, mainly regarding input/output, general processing steps, and general operating modes. To help in a unified experience between all of Gnuastro’s programs, these common operations have the same names and defined in Section 4.1.2 [Common options], page 156.

As in all Gnuastro programs, options can also be given to Segment in configuration files. For a thorough description of Gnuastro’s configuration file parsing, please see Section 4.2 [Configuration files], page 169. All of Segment’s options with a short description are also always available on the command-line with the --help option, see Section 4.3 [Getting help], page 172. To inspect the option values without actually running Segment, append your command with --printparams (or -P).

To help in easy navigation between Segment’s options, they are separately discussed in
the three sub-sections below: Section 7.3.1.1 [Segment input], page 383, discusses how you
can customize the inputs to Segment. Section 7.3.1.2 [Segmentation options], page 387, is
devoted to options specific to the high-level segmentation process. Finally, in Section 7.3.1.3 [Segment output], page 390, we’ll discuss options that affect Segment’s output.

### 7.3.1.1 Segment input

Besides the input dataset (for example astronomical image), Segment also needs to know
the Sky standard deviation and the regions of the dataset that it should segment. The
values dataset is assumed to be Sky subtracted by default. If it isn’t, you can ask Segment
to subtract the Sky internally by calling --sky. For the rest of this discussion, we’ll assume
it is already sky subtracted.

The Sky and its standard deviation can be a single value (to be used for the whole
dataset) or a separate dataset (for a separate value per pixel). If a dataset is used for the
Sky and its standard deviation, they must either be the size of the input image, or have
a single value per tile (generated with --oneelempertile, see Section 4.1.2.2 [Processing
options], page 159, and Section 4.8 [Tessellation], page 189).

The detected regions/pixels can be specified as a detection map (for example see
Section 7.2.2.3 [NoiseChisel output], page 378). If --detection=all, Segment won’t read
any detection map and assume the whole input is a single detection. For example when the dataset is fully covered by a large nearby galaxy/globular cluster.

When dataset are to be used for any of the inputs, Segment will assume they are multiple extensions of a single file by default (when \texttt{--std} or \texttt{--detection} aren't called). For example NoiseChisel's default output Section 7.2.2.3 [NoiseChisel output], page 378. When the Sky-subtracted values are in one file, and the detection and Sky standard deviation are in another, you just need to use \texttt{--detection}: in the absence of \texttt{--std}, Segment will look for both the detection labels and Sky standard deviation in the file given to \texttt{--detection}. Ultimately, if all three are in separate files, you need to call both \texttt{--detection} and \texttt{--std}.

The extensions of the three mandatory inputs can be specified with \texttt{--hdu}, \texttt{--dhdu}, and \texttt{--stdhdu}. For a full discussion on what to give to these options, see the description of \texttt{--hdu} in Section 4.1.2.1 [Input/Output options], page 157. To see their default values (along with all the other options), run Segment with the \texttt{--printparams} (or \texttt{-P}) option. Just recall that in the absence of \texttt{--detection} and \texttt{--std}, all three are assumed to be in the same file. If you only want to see Segment's default values for HDUs on your system, run this command:

\begin{verbatim}
$ astsegment -P | grep hdu
\end{verbatim}

By default Segment will convolve the input with a kernel to improve the signal-to-noise ratio of true peaks. If you already have the convolved input dataset, you can pass it directly to Segment for faster processing (using the \texttt{--convolved} and \texttt{--chdu} options). Just don't forget that the convolved image must also be Sky-subtracted before calling Segment. If a value/file is given to \texttt{--sky}, the convolved values will also be Sky subtracted internally. Alternatively, if you prefer to give a kernel (with \texttt{--kernel} and \texttt{--khdu}), Segment can do the convolution internally. To disable convolution, use \texttt{--kernel=none}.

\textbf{--sky=STR/FLT}

The Sky value(s) to subtract from the input. This option can either be given a constant number or a file name containing a dataset (multiple values, per pixel or per tile). By default, Segment will assume the input dataset is Sky subtracted, so this option is not mandatory.

If the value can't be read as a number, it is assumed to be a file name. When the value is a file, the extension can be specified with \texttt{--skyhdu}. When its not a single number, the given dataset must either have the same size as the output or the same size as the tessellation (so there is one pixel per tile, see Section 4.8 [Tessellation], page 189).

When this option is given, its value(s) will be subtracted from the input and the (optional) convolved dataset (given to \texttt{--convolved}) prior to starting the segmentation process.

\textbf{--skyhdu=STR/INT}

The HDU/extension containing the Sky values. This is mandatory when the value given to \texttt{--sky} is not a number. Please see the description of \texttt{--hdu} in Section 4.1.2.1 [Input/Output options], page 157, for the different ways you can identify a special extension.

\textbf{--std=STR/FLT}

The Sky standard deviation value(s) corresponding to the input. The value can either be a constant number or a file name containing a dataset (multiple
values, per pixel or per tile). The Sky standard deviation is mandatory for Segment to operate.

If the value can’t be read as a number, it is assumed to be a file name. When the value is a file, the extension can be specified with --skyhdu. When its not a single number, the given dataset must either have the same size as the output or the same size as the tessellation (so there is one pixel per tile, see Section 4.8 [Tessellation], page 189).

When this option is not called, Segment will assume the standard deviation is a dataset and in a HDU/extension (--stdhdu) of another one of the input file(s). If a file is given to --detection, it will assume that file contains the standard deviation dataset, otherwise, it will look into input filename (the main argument, without any option).

--stdhdu=INT/STR
The HDU/extension containing the Sky standard deviation values, when the value given to --std is a file name. Please see the description of --hdu in Section 4.1.2.1 [Input/Output options], page 157, for the different ways you can identify a special extension.

--variance
The input Sky standard deviation value/dataset is actually variance. When this option is called, the square root of input Sky standard deviation (see --std) is used internally, not its raw value(s).

-d FITS
--detection=FITS
Detection map to use for segmentation. If given a value of all, Segment will assume the whole dataset must be segmented, see below. If a detection map is given, the extension can be specified with --dhdu. If not given, Segment will assume the desired HDU/extension is in the main input argument (input file specified with no option).

The final segmentation (clumps or objects) will only be over the non-zero pixels of this detection map. The dataset must have the same size as the input image. Only datasets with an integer type are acceptable for the labeled image, see Section 4.5 [Numeric data types], page 178. If your detection map only has integer values, but it is stored in a floating point container, you can use Gnuastro’s Arithmetic program (see Section 6.2 [Arithmetic], page 272) to convert it to an integer container, like the example below:

$ astarithmetic float.fits int32 --output=int.fits

It may happen that the whole input dataset is covered by signal, for example when working on parts of the Andromeda galaxy, or nearby globular clusters (that cover the whole field of view). In such cases, segmentation is necessary over the complete dataset, not just specific regions (detections). By default Segment will first use the undetected regions as a reference to find the proper signal-to-noise ratio of “true” clumps (give a purity level specified with --snquant). Therefore, in such scenarios you also need to manually give a “true” clump signal-to-noise ratio with the --clumpsnthresh option to disable looking into the undetected regions, see Section 7.3.1.2 [Segmentation options], page 387.
In such cases, it is possible to make a detection map that only has the value 1 for all pixels (for example using Section 6.2 [Arithmetic], page 272), but for convenience, you can also use \texttt{--detection=all}.

\texttt{--dhdu} The HDU/extension containing the detection map given to \texttt{--detection}. Please see the description of \texttt{--hdu} in Section 4.1.2.1 [Input/Output options], page 157, for the different ways you can identify a special extension.

\texttt{-k FITS} \texttt{--kernel=FITS}

The name of file containing kernel that will be used to convolve the input image. The usage of this option is identical to NoiseChisel's \texttt{--kernel} option (Section 7.2.2.1 [NoiseChisel input], page 367). Please see the descriptions there for more. To disable convolution, you can give it a value of \texttt{none}.

\texttt{--khdu} The HDU/extension containing the kernel used for convolution. For acceptable values, please see the description of \texttt{--hdu} in Section 4.1.2.1 [Input/Output options], page 157.

\texttt{--convolved=FITS}

The convolved image's file name to avoid internal convolution by Segment. The usage of this option is identical to NoiseChisel's \texttt{--convolved} option. Please see Section 7.2.2.1 [NoiseChisel input], page 367, for a thorough discussion of the usefulness and best practices of using this option.

If you want to use the same convolution kernel for detection (with Section 7.2 [NoiseChisel], page 362) and segmentation, with this option, you can use the same convolved image (that is also available in NoiseChisel) and avoid two convolutions. However, just be careful to use the input to NoiseChisel as the input to Segment also, then use the \texttt{--sky} and \texttt{--std} to specify the Sky and its standard deviation (from NoiseChisel's output). Recall that when NoiseChisel is not called with \texttt{--rawoutput}, the first extension of NoiseChisel's output is the Sky-subtracted input (see Section 7.2.2.3 [NoiseChisel output], page 378). So if you use the same convolved image that you fed to NoiseChisel, but use NoiseChisel's output with Segment's \texttt{--convolved}, then the convolved image won't be Sky subtracted.

\texttt{--chdu} The HDU/extension containing the convolved image (given to \texttt{--convolved}). For acceptable values, please see the description of \texttt{--hdu} in Section 4.1.2.1 [Input/Output options], page 157.

\texttt{-L INT[,INT]} \texttt{--largetilesize=INT[,INT]}

The size of the large tiles to use for identifying the clump S/N threshold over the undetected regions. The usage of this option is identical to NoiseChisel's \texttt{--largetilesize} option (Section 7.2.2.1 [NoiseChisel input], page 367). Please see the descriptions there for more.

The undetected regions can be a significant fraction of the dataset and finding clumps requires sorting of the desired regions, which can be slow. To speed up the processing, Segment finds clumps in the undetected regions over separate large tiles. This allows it to have to sort a much smaller set of pixels and also
to treat them independently and in parallel. Both these issues greatly speed it up. Just be sure to not decrease the large tile sizes too much (less than 100 pixels in each dimension). It is important for them to be much larger than the clumps.

### 7.3.1.2 Segmentation options

The options below can be used to configure every step of the segmentation process in the Segment program. For a more complete explanation (with figures to demonstrate each step), please see Section 3.2 of Akhlaghi and Ichikawa [2015] ([https://arxiv.org/abs/1505.01664](https://arxiv.org/abs/1505.01664)), and also Section 7.3 [Segment], page 381. By default, Segment will follow the procedure described in the paper to find the S/N threshold based on the noise properties. This can be disabled by directly giving a trustable signal-to-noise ratio to the `--clumpsnthresh` option.

Recall that you can always see the full list of Gnuastro’s options with the `--help` (see Section 4.3 [Getting help], page 172), or `--printparams` (or `-P`) to see their values (see Section 4.1.2.3 [Operating mode options], page 162).

- **-B FLT**
  - `--minskyfrac=FLT`
    - Minimum fraction (value between 0 and 1) of Sky (undetected) areas in a large tile. Only (large) tiles with a fraction of undetected pixels (Sky) greater than this value will be used for finding clumps. The clumps found in the undetected areas will be used to estimate a S/N threshold for true clumps. Therefore this is an important option (to decrease) in crowded fields. Operationally, this is almost identical to NoiseChisel’s `--minskyfrac` option (Section 7.2.2.2 [Detection options], page 370). Please see the descriptions there for more.

- **--minima**
  - Build the clumps based on the local minima, not maxima. By default, clumps are built starting from local maxima (see Figure 8 of Akhlaghi and Ichikawa [2015] ([https://arxiv.org/abs/1505.01664](https://arxiv.org/abs/1505.01664))). Therefore, this option can be useful when you are searching for true local minima (for example absorption features).

- **-m INT**
  - `--snminarea=INT`
    - The minimum area which a clump in the undetected regions should have in order to be considered in the clump Signal to noise ratio measurement. If this size is set to a small value, the Signal to noise ratio of false clumps will not be accurately found. It is recommended that this value be larger than the value to NoiseChisel’s `--snminarea`. Because the clumps are found on the convolved (smoothed) image while the pseudo-detections are found on the input image. You can use `--checksn` and `--checksegmentation` to see if your chosen value is reasonable or not.

- **--checksn**
  - Save the S/N values of the clumps over the sky and detected regions into separate tables. If `--tableformat` is a FITS format, each table will be written into a separate extension of one file suffixed with `_clumpsn.fits`. If it is plain text, a separate file will be made for each table (ending in `_clumpsn_sky.txt`
and _clumpsn_det.txt). For more on --tableformat see Section 4.1.2.1 [Input/Output options], page 157.

You can use these tables to inspect the S/N values and their distribution (in combination with the --checksegmentation option to see where the clumps are). You can use Gnuastro’s Section 7.1 [Statistics], page 337, to make a histogram of the distribution (ready for plotting in a text file, or a crude ASCII-art demonstration on the command-line).

With this option, Segment will abort as soon as the two tables are created. This allows you to inspect the steps leading to the final S/N quantile threshold, this behavior can be disabled with --continueaftercheck.

--minnumfalse=INT
The minimum number of clumps over undetected (Sky) regions to identify the requested Signal-to-Noise ratio threshold. Operationally, this is almost identical to NoiseChisel’s --minnumfalse option (Section 7.2.2.2 [Detection options], page 370). Please see the descriptions there for more.

-c FLT
--snquant=FLT
The quantile of the signal-to-noise ratio distribution of clumps in undetected regions, used to define true clumps. After identifying all the usable clumps in the undetected regions of the dataset, the given quantile of their signal-to-noise ratios is used to define the signal-to-noise ratio of a “true” clump. Effectively, this can be seen as an inverse p-value measure. See Figure 9 and Section 3.2.1 of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664) for a complete explanation. The full distribution of clump signal-to-noise ratios over the undetected areas can be saved into a table with --checksn option and visually inspected with --checksegmentation.

-v
--keepmaxnearriver
Keep a clump whose maximum (minimum if --minima is called) flux is 8-connected to a river pixel. By default such clumps over detections are considered to be noise and are removed irrespective of their brightness (see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394). Over large profiles, that sink into the noise very slowly, noise can cause part of the profile (which was flat without noise) to become a very large and with a very high Signal to noise ratio. In such cases, the pixel with the maximum flux in the clump will be immediately touching a river pixel.

-s FLT
--clumpsnthresh=FLT
The signal-to-noise threshold for true clumps. If this option is given, then the segmentation options above will be ignored and the given value will be directly used to identify true clumps over the detections. This can be useful if you have a large dataset with similar noise properties. You can find a robust signal-to-noise ratio based on a (sufficiently large) smaller portion of the dataset. Afterwards, with this option, you can speed up the processing on the whole dataset. Other
scenarios where this option may be useful is when, the image might not contain enough/any Sky regions.

-G FLT
--gthresh=FLT
Threshold (multiple of the sky standard deviation added with the sky) to stop growing true clumps. Once true clumps are found, they are set as the basis to segment the detected region. They are grown until the threshold specified by this option.

-y INT
--minriverlength=INT
The minimum length of a river between two grown clumps for it to be considered in signal-to-noise ratio estimations. Similar to --snminarea, if the length of the river is too short, the signal-to-noise ratio can be noisy and unreliable. Any existing rivers shorter than this length will be considered as non-existent, independent of their Signal to noise ratio. The clumps are grown on the input image, therefore this value can be smaller than the value given to --snminarea. Recall that the clumps were defined on the convolved image so --snminarea should be larger.

-O FLT
--objbordersn=FLT
The maximum Signal to noise ratio of the rivers between two grown clumps in order to consider them as separate ‘objects’. If the Signal to noise ratio of the river between two grown clumps is larger than this value, they are defined to be part of one ‘object’. Note that the physical reality of these ‘objects’ can never be established with one image, or even multiple images from one broadband filter. Any method we devise to define ‘object’s over a detected region is ultimately subjective.

Two very distant galaxies or satellites in one halo might lie in the same line of sight and be detected as clumps on one detection. On the other hand, the connection (through a spiral arm or tidal tail for example) between two parts of one galaxy might have such a low surface brightness that they are broken up into multiple detections or objects. In fact if you have noticed, exactly for this purpose, this is the only Signal to noise ratio that the user gives into NoiseChisel. The ‘true’ detections and clumps can be objectively identified from the noise characteristics of the image, so you don’t have to give any hand input Signal to noise ratio.

--checksegmentation
A file with the suffix _seg.fits will be created. This file keeps all the relevant steps in finding true clumps and segmenting the detections into multiple objects in various extensions. Having read the paper or the steps above. Examining this file can be an excellent guide in choosing the best set of parameters. Note that calling this function will significantly slow NoiseChisel. In verbose mode (without the --quiet option, see Section 4.1.2.3 [Operating mode options], page 162) the important steps (along with their extension names) will also be reported.
With this option, NoiseChisel will abort as soon as the two tables are created. This behavior can be disabled with `--continueaftercheck`.

7.3.1.3 Segment output

The main output of Segment are two label datasets (with integer types, separating the dataset’s elements into different classes). They have HDU/extension names of `CLUMPS` and `OBJECTS`.

Similar to all Gnuastro’s FITS outputs, the zero-th extension/HDU of the main output file only contains header keywords and image or table. It contains the Segment input files and parameters (option names and values) as FITS keywords. Note that if an option name is longer than 8 characters, the keyword name is the second word. The first word is `HIERARCH`. Also note that according to the FITS standard, the keyword names must be in capital letters, therefore, if you want to use Grep to inspect these keywords, use the `-i` option, like the example below.

```
$ astfits image_segmented.fits -h0 | grep -i snquant
```

By default, besides the `CLUMPS` and `OBJECTS` extensions, Segment’s output will also contain the (technically redundant) input dataset and the sky standard deviation dataset (if it wasn’t a constant number). This can help in visually inspecting the result when viewing the images as a “Multi-extension data cube” in SAO DS9 for example (see Section 10.4 [Viewing FITS file contents with DS9 or TOPCAT], page 494). You can simply flip through the extensions and see the same region of the image and its corresponding clumps/object labels. It also makes it easy to feed the output (as one file) into MakeCatalog when you intend to make a catalog afterwards (see Section 7.4 [MakeCatalog], page 392). To remove these redundant extensions from the output (for example when designing a pipeline), you can use `--rawoutput`.

The `OBJECTS` and `CLUMPS` extensions can be used as input into Section 7.4 [MakeCatalog], page 392, to generate a catalog for higher-level analysis. If you want to treat each clump separately, you can give a very large value (or even a NaN, which will always fail) to the `--gthresh` option (for example `--gthresh=1e10` or `--gthresh=nan`), see Section 7.3.1.2 [Segmentation options], page 387.

For a complete definition of clumps and objects, please see Section 3.2 of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664) and Section 7.3.1.2 [Segmentation options], page 387. The clumps are “true” local maxima (minima if `--minima` is called) and their surrounding pixels until a local minimum/maximum (caused by noise fluctuations, or another “true” clump). Therefore it may happen that some of the input detections aren’t covered by clumps at all (very diffuse objects without any strong peak), while some objects may contain many clumps. Even in those that have clumps, there will be regions that are too diffuse. The diffuse regions (within the input detected regions) are given a negative label (-1) to help you separate them from the undetected regions (with a value of zero).

Each clump is labeled with respect to its host object. Therefore, if an object has three clumps for example, the clumps within it have labels 1, 2 and 3. As a result, if an initial detected region has multiple objects, each with a single clump, all the clumps will have a label of 1. The total number of clumps in the dataset is stored in the `NCLUMPS` keyword of the `CLUMPS` extension and printed in the verbose output of Segment (when `--quiet` is not called).
The **OBJECTS** extension of the output will give a positive counter/label to every detected pixel in the input. As described in Akhlaghi and Ichikawa [2015], the true clumps are grown until a certain threshold. If the grown clumps touch other clumps and the connection is strong enough, they are considered part of the same **object**. Once objects (grown clumps) are identified, they are grown to cover the whole detected area.

The options to configure the output of Segment are listed below:

---

**--continueaftercheck**

Don’t abort Segment after producing the check image(s). The usage of this option is identical to NoiseChisel’s **--continueaftercheck** option (Section 7.2.2.1 [NoiseChisel input], page 367). Please see the descriptions there for more.

**--noobjects**

Abort Segment after finding true clumps and don’t continue with finding options. Therefore, no **OBJECTS** extension will be present in the output. Each true clump in **CLUMPS** will get a unique label, but diffuse regions will still have a negative value.

To make a catalog of the clumps, the input detection map (where all the labels are one) can be fed into Section 7.4 [MakeCatalog], page 392, along with the input detection map to Segment (that only had a value of 1 for all detected pixels) with **--clumpscat**. In this way, MakeCatalog will assume all the clumps belong to a single “object”.

**--grownclumps**

In the output **CLUMPS** extension, store the grown clumps. If a detected region contains no clumps or only one clump, then it will be fully given a label of 1 (no negative valued pixels).

**--rawoutput**

Only write the **CLUMPS** and **OBJECTS** datasets in the output file. Without this option (by default), the first and last extensions of the output will the Sky-subtracted input dataset and the Sky standard deviation dataset (if it wasn’t a number). When the datasets are small, these redundant extensions can make it convenient to inspect the results visually or feed the output to Section 7.4 [MakeCatalog], page 392, for measurements. Ultimately both the input and Sky standard deviation datasets are redundant (you had them before running Segment). When the inputs are large/numerous, these extra dataset can be a burden.

---

**Save space:** with the **--rawoutput**, Segment’s output will only be two labeled datasets (only containing integers). Since they have no noise, such datasets can be compressed very effectively (without any loss of data) with exceptionally high compression ratios. You can use the following command to compress it with the best ratio:

```
$ gzip --best segment_output.fits
```

The resulting `.fits.gz` file can then be fed into any of Gnuastro’s programs directly, without having to decompress it separately (it will just take them a little longer, because they have to decompress it internally before use).
When the input is a 2D image, to inspect NoiseChisel’s output you can configure SAO DS9 in your Graphic User Interface (GUI) to open NoiseChisel’s output as a multi-extension data cube. This will allow you to flip through the different extensions and visually inspect the results. This process has been described for the GNOME GUI (most common GUI in GNU/Linux operating systems) in Section 10.4 [Viewing FITS file contents with DS9 or TOPCAT], page 494.

7.4 MakeCatalog

At the lowest level, a dataset (for example an image) is just a collection of values, placed after each other in any number of dimensions (for example an image is a 2D dataset). Each data-element (pixel) just has two properties: its position (relative to the rest) and its value. In higher-level analysis, an entire dataset (an image for example) is rarely treated as a singular entity. You usually want to know/measure the properties of the (separate) scientifically interesting targets that are embedded in it. For example the magnitudes, positions and elliptical properties of the galaxies that are in the image.

MakeCatalog is Gnuastro’s program for localized measurements over a dataset. In other words, MakeCatalog is Gnuastro’s program to convert low-level datasets (like images), to high level catalogs. The role of MakeCatalog in a scientific analysis and the benefits of its model (where detection/segmentation is separated from measurement) is discussed in Akhlaghi [2016] (https://arxiv.org/abs/1611.06387v1) and summarized in Section 7.4.1 [Detection and catalog production], page 393. We strongly recommend reading this short paper for a better understanding of this methodology. Understanding the effective usage of MakeCatalog, will thus also help effective use of other (lower-level) Gnuastro’s programs like Section 7.2 [NoiseChisel], page 362, or Section 7.3 [Segment], page 381.

It is important to define your regions of interest for measurements before running MakeCatalog. MakeCatalog is specialized in doing measurements accurately and efficiently. Therefore MakeCatalog will not do detection, segmentation, or defining apertures on requested positions in your dataset. Following Gnuastro’s modularity principle, there are separate and highly specialized and customizable programs in Gnuastro for these other jobs as shown below (for a usage example in a real-world analysis, see Section 2.2 [General program usage tutorial], page 29, and Section 2.3 [Detecting large extended targets], page 76).

- Section 6.2 [Arithmetic], page 272: Detection with a simple threshold.
- Section 7.2 [NoiseChisel], page 362: Advanced detection.
- Section 7.3 [Segment], page 381: Segmentation (substructure over detections).
- Section 8.1 [MakeProfiles], page 440: Aperture creation for known positions.

These programs will/can return labeled dataset(s) to be fed into MakeCatalog. A labeled dataset for measurement has the same size/dimensions as the input, but with integer valued pixels that have the label/counter for each sub-set of pixels that must be measured together. For example all the pixels covering one galaxy in an image, get the same label.

12 You can derive the over-all properties of a complete dataset (1D table column, 2D image, or 3D data-cube) treated as a single entity with Gnuastro’s Statistics program (see Section 7.1 [Statistics], page 337).

13 A published paper cannot undergo any more change, so this manual is the definitive guide.
The requested measurements are then done on similarly labeled pixels. The final result is a catalog where each row corresponds to the measurements on pixels with a specific label. For example, the flux weighted average position of all the pixels with a label of 42 will be written into the 42nd row of the output catalog/table’s central position column. Similarly, the sum of all these pixels will be the 42nd row in the brightness column, etc. Pixels with labels equal to, or smaller than, zero will be ignored by MakeCatalog. In other words, the number of rows in MakeCatalog’s output is already known before running it (the maximum value of the labeled dataset).

Before getting into the details of running MakeCatalog (in Section 7.4.6 [Invoking MakeCatalog], page 408, we’ll start with a discussion on the basics of its approach to separating detection from measurements in Section 7.4.1 [Detection and catalog production], page 393. A very important factor in any measurement is understanding its validity range, or limits. Therefore in Section 7.4.3 [Quantifying measurement limits], page 398, we’ll discuss how to estimate the reliability of the detection and basic measurements. This section will continue with a derivation of elliptical parameters from the labeled datasets in Section 7.4.4 [Measuring elliptical parameters], page 404. For those who feel MakeCatalog’s existing measurements/columns aren’t enough and would like to add further measurements, in Section 7.4.5 [Adding new columns to MakeCatalog], page 407, a checklist of steps is provided for readily adding your own new measurements/columns.

### 7.4.1 Detection and catalog production

Most existing common tools in low-level astronomical data-analysis (for example SExtractor) merge the two processes of detection and measurement (catalog production) in one program. However, in light of Gnuastro’s modularized approach (modeled on the Unix system) detection is separated from measurements and catalog production. This modularity is therefore new to many experienced astronomers and deserves a short review here. Further discussion on the benefits of this methodology can be seen in Akhlaghi [2016] (https://arxiv.org/abs/1611.06387v1).

As discussed in the introduction of Section 7.4 [MakeCatalog], page 392, detection (identifying which pixels to do measurements on) can be done with different programs. Their outputs (a labeled dataset) can be directly fed into MakeCatalog to do the measurements and write the result as a catalog/table. Beyond that, Gnuastro’s modular approach has many benefits that will become clear as you get more experienced in astronomical data analysis and want to be more creative in using your valuable data for the exciting scientific project you are working on. In short the reasons for this modularity can be classified as below:

- **Simplicity/robustness of independent, modular tools:** making a catalog is a logically separate process from labeling (detection, segmentation, or aperture production). A user might want to do certain operations on the labeled regions before creating a catalog for them. Another user might want the properties of the same pixels/objects in another image (another filter for example) to measure the colors or SED fittings.

Here is an example of doing both: suppose you have images in various broad band filters at various resolutions and orientations. The image of one color will thus not lie

---

14 See Section 7.4.4 [Measuring elliptical parameters], page 404, for a discussion on this and the derivation of positional parameters, which includes the center.

15 https://www.astromatic.net/software/sextractor
exactly on another or even be in the same scale. However, it is imperative that the
same pixels be used in measuring the colors of galaxies.
To solve the problem, NoiseChisel can be run on the reference image to generate the
labeled detection image. Afterwards, the labeled image can be warped into the grid of
the other color (using Section 6.4 [Warp], page 328). MakeCatalog will then generate
the same catalog for both colors (with the different labeled images). It is currently
customary to warp the images to the same pixel grid, however, modification of the
scientific dataset is very harmful for the data and creates correlated noise. It is much
more accurate to do the transformations on the labeled image.
• Complexity of a monolith: Adding in a catalog functionality to the detector program
will add several more steps (and many more options) to its processing that can equally
well be done outside of it. This makes following what the program does harder for the
users and developers, it can also potentially add many bugs.
As an example, if the parameter you want to measure over one profile is not provided
by the developers of MakeCatalog. You can simply open this tiny little program and
add your desired calculation easily. This process is discussed in Section 7.4.5 [Adding
new columns to MakeCatalog], page 407. However, if making a catalog was part of
NoiseChisel for example, adding a new column/measurement would require a lot of
energy to understand all the steps and internal structures of that huge program. It
might even be so intertwined with its processing, that adding new columns might cause
problems/bugs in its primary job (detection).

7.4.2 Brightness, Flux, Magnitude and Surface brightness
Astronomical data pixels are usually in units of counts\textsuperscript{16} or electrons or either one divided
by seconds. To convert from the counts to electrons, you will need to know the instrument
gain. In any case, they can be directly converted to energy or energy/time using the basic
hardware (telescope, camera and filter) information (that is summarized in the zero point,
and we will discuss below). We will continue the discussion assuming the pixels are in units
of energy/time.

**Brightness** The *brightness* of an object is defined as its total detected energy per time.
In the case of an imaged source, this is simply the sum of the pixels that are
associated with that detection by our detection tool (for example Section 7.2
[NoiseChisel], page 362\textsuperscript{17}). The *flux* of an object is defined in units of
energy/time/collection-area. For an astronomical target, the flux is therefore
defined as its brightness divided by the area used to collect the light from
the source: or the telescope aperture (for example in units of $cm^2$). Knowing
the flux ($f$) and distance to the object ($r$), we can define its *luminosity*:
$L = 4\pi r^2 f$.
Therefore, while flux and luminosity are intrinsic properties of the object,
brightness depends on our detecting tools (hardware and software). In low-
level observational astronomy data analysis, we are usually more concerned
with measuring the brightness, because it is the thing we directly measure from

\textsuperscript{16} Counts are also known as analog to digital units (ADU).
\textsuperscript{17} If further processing is done, for example the Kron or Petrosian radii are calculated, then the detected
area is not sufficient and the total area that was within the respective radius must be used.
the image pixels and create in catalogs. On the other hand, luminosity is used in higher-level analysis (after image contents are measured as catalogs to deduce physical interpretations). It is just important avoid possible confusion between luminosity and brightness because both have the same units of energy per seconds.

Magnitude

Images of astronomical objects span over a very large range of brightness. With the Sun (as the brightest object) being roughly $2.5^{60} = 10^{24}$ times brighter than the fainter galaxies we can currently detect in the deepest images. Therefore discussing brightness directly will involve a large range of values which is inconvenient. So astronomers have chosen to use a logarithmic scale to talk about the brightness of astronomical objects.

But the logarithm can only be usable with a dimensionless value that is always positive. Fortunately brightness is always positive (at least in theory). To remove the dimensions, we divide the brightness of the object ($B$) by a reference brightness ($B_r$). We then define a logarithmic scale as magnitude through the relation below. The $-2.5$ factor in the definition of magnitudes is a legacy of our ancient colleagues and in particular Hipparchus of Nicaea (190-120 BC).

$$m - m_r = -2.5 \log_{10} \left( \frac{B}{B_r} \right)$$

$m$ is defined as the magnitude of the object and $m_r$ is the pre-defined magnitude of the reference brightness. For estimating the error in measuring a magnitude, see Section 7.4.3 [Quantifying measurement limits], page 398.

Zero point

A unique situation in the magnitude equation above occurs when the reference brightness is unity ($B_r = 1$). This brightness will thus summarize all the hardware-specific parameters discussed above (like the conversion of pixel values to physical units) into one number. That reference magnitude is commonly known as the Zero point magnitude because when $B = B_r = 1$, the right side of the magnitude definition above will be zero. Using the zero point magnitude ($Z$), we can write the magnitude relation above in a more simpler format:

$$m = -2.5 \log_{10}(B) + Z$$

Having the zero point of an image, you can convert its pixel values to physical units of microJanskys (or $\mu Jy$). This enables direct pixel-based comparisons with images from other instruments. Jansky is a commonly used unit for measuring spectral flux density and one Jansky is equivalent to $10^{-26} W/m^2/Hz$ (watts per square meter per hertz).

---

In practice, for very faint objects, if the background brightness is over-subtracted, we may end up with a negative brightness in a real object.

Comparing data from different instruments assumes instrument and observation signatures are properly corrected, things like the flat-field or the Sky absorption.
This conversion can be done with the fact that in the AB magnitude standard\textsuperscript{20}, 3631 Jy corresponds to the zero-th magnitude, therefore $B \equiv 3631 \times 10^6 \mu$Jy and $m \equiv 0$. We can therefore estimate the brightness ($B_z$, in $\mu$Jy) corresponding to the image zero point ($Z$) using this equation:

$$m - Z = -2.5 \log_{10}(B/B_z)$$

$$0 - Z = -2.5 \log_{10}\left(\frac{3631 \times 10^6}{B_z}\right)$$

$$B_z = 3631 \times 10^{(6-Z/2.5)} \mu Jy$$

Because the image zero point corresponds to a pixel value of 1, the $B_z$ value calculated above also corresponds to a pixel value of 1. Therefore you simply have to multiply your image by $B_z$ to convert it to $\mu$Jy. Don’t forget that this only applies when your zero point was also estimated in the AB magnitude system. On the command-line, you can estimate this value for a certain zero point with AWK, then multiply it to all the pixels in the image with Section 6.2 [Arithmetic], page 272. For example let’s assume you are using an SDSS image with a zero point of 22.5:

```bash
bz=$(echo 22.5 | awk '{print 3631 * 10^(6-$1/2.5)}')
astarithmetic sdss.fits $bz x --output=sdss-in-muJy.fits
```

But in Gnuastro, it gets even easier: Arithmetic has an operator called counts-to-jy. This will directly convert your image pixels (in units of counts) to Janskys though a provided AB Magnitude-based zero point like below. See Section 6.2.3 [Arithmetic operators], page 276, for more.

```bash
$ astarithmetic sdss.fits 22.5 counts-to-jy
```

**Be careful with the exposure time:** As described at the start of this section, we are assuming your data are in units of counts/sec. As a result, the counts you get from the command above, are only for one second of exposure! Please see the discussion below in “Magnitude to counts” for more.

Magnitude to counts (accounting for exposure time)

Until now, we had assumed that the data are in units of counts/sec. As a result, the equations given above (in the “Zero point” item to convert magnitudes to pixel counts), give the count level for the reference (1 second) exposure. But we rarely take 1 second exposures! It is therefore very important to take the exposure time into account in scenarios like simulating observations with varying exposure times (where you need to know how many counts the object of a certain magnitude will add to a certain image with a certain exposure time).

To clarify the concept, let’s define $C$ as the counted electrons (which has a linear relation with the photon energy entering the CCD pixel). In this case, if

\textsuperscript{20} https://en.wikipedia.org/wiki/AB_magnitude
an object of brightness $B$ is observed for $t$ seconds, it will accumulate $C = B \times t$ counts. Therefore, the generic magnitude equation above can be written as:

$$ m = -2.5 \log_{10}(B) + Z = -2.5 \log_{10}(C/t) + Z $$

From this, we can derive $C(t)$ in relation to $C(1)$, or counts from a 1 second exposure, using this relation:

$$ C(t) = t \times 10^{(m-Z)/2.5} = t \times C(1) $$

In other words, you should simply multiply the counts for one second with the number of observed seconds.

Another approach is to shift the time-dependence of the counts into the zero point (after all exposure time is also a hardware issue). Let’s derive the equation below:

$$ m = -2.5 \log_{10}(C/t) + Z = -2.5 \log_{10}(C) + 2.5 \log_{10}(t) + Z $$

Therefore, defining an exposure-time-dependent zero point as $Z(t)$, we can directly correlate a certain object’s magnitude with counts after an exposure of $t$ seconds:

$$ m = -2.5 \log_{10}(C) + Z(t) \quad \text{where} \quad Z(t) = Z + 2.5 \log_{10}(t) $$

This solution is useful in programs like Section 7.4 [MakeCatalog], page 392, or Section 8.1 [MakeProfiles], page 440, when you can’t (or don’t want to: because of the extra storage/speed costs) manipulate the values image (for example divide it by the exposure time to use a counts/sec zero point).

**Surface brightness**

Another important concept is the distribution of an object’s brightness over its area. For this, we define the *surface brightness* to be the magnitude of an object’s brightness divided by its solid angle over the celestial sphere (or coverage in the sky, commonly in units of arcsec$^2$). The solid angle is expressed in units of arcsec$^2$ because astronomical targets are usually much smaller than one steradian. Recall that the steradian is the dimension-less SI unit of a solid angle and 1 steradian covers $1/4\pi$ (almost 8%) of the full celestial sphere.

Surface brightness is therefore most commonly expressed in units of mag/arcsec2. For example when the brightness is measured over an area of $A$ arcsec$^2$, then the surface brightness becomes:

$$ S = -2.5 \log_{10}(B/A) + Z = -2.5 \log_{10}(B) + 2.5 \log_{10}(A) + Z $$

In other words, the surface brightness (in units of mag/arcsec$^2$) is related to the object’s magnitude ($m$) and area ($A$, in units of arcsec$^2$) through this equation:

---

21 Recall that counts another name for ADUs, which already includes the CCD gain.
\[ S = m + 2.5 \log_{10}(A) \]

A common mistake is to follow the mag/arcsec\(^2\) unit literally, and divide the object’s magnitude by its area. But this is wrong because magnitude is a logarithmic scale while area is linear. It is the brightness that should be divided by the solid angle because both have linear scales. The magnitude of that ratio is then defined to be the surface brightness.

A common scenario is to convert an image’s pixel values to surface brightness, when you know its zeropoint. This can be done with the two simple commands below. First, we derive the pixel area (in arcsec\(^2\)) then we use Arithmetic to convert the pixels into surface brightness, see below for the details.

```bash
$ pixarea=$(astfits image.fits --pixelscale --quiet |
    awk '{print $1*3600*$2*3600}')
$ astarithmetic image.fits abs 26.23 counts-to-mag
    $pixarea log10 2.5 x +
    --output=image-sb.fits
```

Within the Arithmetic call, first we apply the `abs` operator to convert all negative pixels to positive. This is done because magnitudes are defined in log scale, which is not defined in the negative. In a properly reduced image, negative pixels will be only due to noise and their fluctuations will be similar to positive noise pixels, so this is no problem. The result is then fed into Arithmetic’s `counts-to-mag` operator with the zeropoint which will convert all pixels to magnitudes. Finally we calculate the extra \(2.5 \log_{10}(A)\) term (with the `$pixarea log10 2.5 x` term) and add it (with `+`) to each pixel of the magnitude image to have the pixel’s surface brightness. See Section 6.2.1 [Reverse polish notation], page 272, for more on Arithmetic’s notation and Section 6.2.3 [Arithmetic operators], page 276, for a description of each operator.

**Don’t warp or convolve magnitude or surface brightness images:** Warping an image involves calculating new pixel values (of the new pixel grid) from the old pixel values. Convolution is also a process of finding the weighted mean of pixel values. During these processes, many arithmetic operations are done on the original pixel values, for example addition or multiplication. However, \(\log_{10}(a + b) \neq \log_{10}(a) + \log_{10}(b)\). Therefore after calculating a magnitude or surface brightness image, do not apply any such operations on it! If you need to warp or convolve the image, do it **before** the conversion.

### 7.4.3 Quantifying measurement limits

No measurement on a real dataset can be perfect: you can only reach a certain level/limit of accuracy and a meaningful (scientific) analysis requires an understanding of these limits. Different datasets have different noise properties and different detection methods (one method/algorith/software that is run with a different set of parameters is considered as a different detection method) will have different abilities to detect or measure certain kinds
of signal (astronomical objects) and their properties in the dataset. Hence, quantifying the
detection and measurement limitations with a particular dataset and analysis tool is the
most crucial/critical aspect of any high-level analysis. In two separate tutorials, we have
touched upon some of these points. So to see the discussions below in action (on real data),
see Section 2.2.14 [Measuring the dataset limits], page 54, and Section 2.3.4 [Image surface
brightness limit], page 88.

Here, we’ll review some of the most commonly used methods to quantify the limits in
astronomical data analysis and how MakeCatalog makes it easy to measure them. De-
pending on the higher-level analysis, there are more tests that must be done, but these are
relatively low-level and usually necessary in most cases. In astronomy, it is common to use
the magnitude (a unit-less scale) and physical units, see Section 7.4.2 [Brightness, Flux,
Magnitude and Surface brightness], page 394. Therefore the measurements discussed here
are commonly used in units of magnitudes.

7.4.3.1 Magnitude measurement error of each detection

The raw error in measuring the magnitude is only meaningful when the object’s magni-
tude is brighter than the upper-limit magnitude (see below). As discussed in Section 7.4.2
[Brightness, Flux, Magnitude and Surface brightness], page 394, the magnitude \( M \) of an
object with brightness \( B \) and zero point magnitude \( z \) can be written as:

\[
M = -2.5 \log_{10}(B) + z
\]

Calculating the derivative with respect to \( B \), we get:

\[
\frac{dM}{dB} = -2.5 \times \frac{1}{B \times \ln(10)}
\]

From the Tailor series \( \Delta M = dM/dB \times \Delta B \), we can write:

\[
\Delta M = \left| \frac{-2.5}{\ln(10)} \right| \times \frac{\Delta B}{B}
\]

But, \( \Delta B/B \) is just the inverse of the Signal-to-noise ratio \( (S/N) \), so we can write the error
in magnitude in terms of the signal-to-noise ratio:

\[
\Delta M = \frac{2.5}{S/N \times \ln(10)}
\]

MakeCatalog uses this relation to estimate the magnitude errors. The signal-to-noise ra-
tio is calculated in different ways for clumps and objects (see Akhlaghi and Ichikawa [2015]
(https://arxiv.org/abs/1505.01664)), but this single equation can be used to estimate
the measured magnitude error afterwards for any type of target.
7.4.3.2 Surface brightness error of each detection

We can derive the error in measuring the surface brightness based on the surface brightness (SB) equation of Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394, and the generic magnitude error ($\Delta M$) of Section 7.4.3.1 [Magnitude measurement error of each detection], page 399. Let’s set $A$ to represent the area and $\Delta A$ to represent the error in measuring the area. For more on $\Delta A$, see the description of --spatialresolution in Section 7.4.6.1 [MakeCatalog inputs and basic settings], page 409.

$$\Delta(SB) = \Delta M + \frac{-2.5}{\ln(10)} \times \frac{\Delta A}{A}$$

In the surface brightness equation mentioned above, $A$ is in units of arcsecond squared and the conversion between arcseconds to pixels is a multiplication factor. Therefore as long as $A$ and $\Delta A$ have the same units, it doesn’t matter if they are in arcseconds or pixels. Since the measure of spatial resolution (or area error) is the FWHM of the PSF which is usually defined in terms of pixels, its more intuitive to use pixels for $A$ and $\Delta A$.

7.4.3.3 Completeness limit of each detection

As the surface brightness of the objects decreases, the ability to detect them will also decrease. An important statistic is thus the fraction of objects of similar morphology and brightness that will be detected with our detection algorithm/parameters in a given image. This fraction is known as completeness. For brighter objects, completeness is 1: all bright objects that might exist over the image will be detected. However, as we go to objects of lower overall surface brightness, we will fail to detect a fraction of them, and fainter than a certain surface brightness level (for each morphology), nothing will be detectable in the image: you will need more data to construct a “deeper” image. For a given profile and dataset, the magnitude where the completeness drops below a certain level (usually above 90%) is known as the completeness limit.

Another important parameter in measuring completeness is purity: the fraction of true detections to all true detections. In effect purity is the measure of contamination by false detections: the higher the purity, the lower the contamination. Completeness and purity are anti-correlated: if we can allow a large number of false detections (that we might be able to remove by other means), we can significantly increase the completeness limit.

One traditional way to measure the completeness and purity of a given sample is by embedding mock profiles in regions of the image with no detection. However in such a study we must be really careful to choose model profiles as similar to the target of interest as possible.

7.4.3.4 Upper limit magnitude of each detection

Due to the noisy nature of data, it is possible to get arbitrarily low values for a faint object’s brightness (or arbitrarily high magnitudes). Given the scatter caused by the dataset’s noise, values fainter than a certain level are meaningless: another similar depth observation will give a radically different value.

For example, assume that you have done your detection and segmentation on one filter and now you do measurements over the same labeled regions, but on other filters to measure colors (as we did in the tutorial Section 2.2.13 [Segmentation and making a catalog],
Some objects are not going to have any significant signal in the other filters, but for example, you measure magnitude of 36 for one of them! This is clearly unreliable (no dataset in current astronomy is able to detect such a faint signal). In another image with the same depth, using the same filter, you might measure a magnitude of 30 for it, and yet another might give you 33. Furthermore, the total brightness might actually be negative in some images of the same depth (due to noise). In these cases, no magnitude can be defined and MakeCatalog will place a NaN there (recall that a magnitude is a base-10 logarithm).

Using such unreliable measurements will directly affect our analysis, so we must not use the raw measurements. When approaching the limits of your detection method, it is therefore important to be able to identify such cases. But how can we know how reliable a measurement of one object on a given dataset is?

When we confront such unreasonably faint magnitudes, there is one thing we can deduce: that if something actually exists under our labeled pixels (possibly buried deep under the noise), it’s inherent magnitude is fainter than an upper limit magnitude. To find this upper limit magnitude, we place the object’s footprint (segmentation map) over a random part of the image where there are no detections, and measure the total brightness within the footprint. Doing this a large number of times will give us a distribution of brightness values. The standard deviation ($\sigma$) of that distribution can be used to quantify the upper limit magnitude for that particular object (given its particular shape and area):

$$M_{\text{up,}n\sigma} = -2.5 \times \log_{10}(n\sigma_m) + z \quad [\text{mag/target}]$$

Traditionally, faint/small object photometry was done using fixed circular apertures (for example with a diameter of $N$ arc-seconds) and there wasn’t much processing involved (to make a deep stack). Hence, the upper limit was synonymous with the surface brightness limit discussed above: one value for the whole image. The problem with this simplified approach is that the number of pixels in the aperture directly affects the final distribution and thus magnitude. Also the image correlated noise might actually create certain patterns, so the shape of the object can also affect the final result. Fortunately, with the much more advanced hardware and software of today, we can make customized segmentation maps (footprint) for each object and have enough computing power to actually place that footprint over many random places. As a result, the per-target upper-limit magnitude and general surface brightness limit have diverged.

When any of the upper-limit-related columns requested, MakeCatalog will randomly place each target’s footprint over the undetected parts of the dataset as described above, and estimate the required properties. The procedure is fully configurable with the options in Section 7.4.6.2 [Upper-limit settings], page 413. You can get the full list of upper-limit related columns of MakeCatalog with this command (the extra -- before --upperlimit is necessary):

```bash
$ astmkcatalog --help | grep -- --upperlimit
```

\[22\] Without the extra --, grep will assume that --upperlimit is one of its own options, and will thus abort, complaining that it has no option with this name.
7.4.3.5 Magnitude limit of image

Suppose we have taken two images of the same field of view with the same CCD, once with a smaller telescope, and once with a larger one. Because we used the same CCD, the noise will be very similar. However, the larger telescope has gathered more light, therefore the same star or galaxy will have a higher signal-to-noise ratio (S/N) in the image taken with the larger one. The same applies for a stacked image of the field compared to a single-exposure image of the same telescope.

This concept is used by some researchers to define the “magnitude limit” or “detection limit” at a certain S/N (sometimes 10, 5 or 3 for example, also written as $10\sigma$, $5\sigma$ or $3\sigma$). To do this, they measure the magnitude and signal-to-noise ratio of all the objects within an image and measure the mean (or median) magnitude of objects at the desired S/N. A fully working example of deriving the magnitude limit is available in the tutorials section: Section 2.2.14 [Measuring the dataset limits], page 54.

However, this method should be used with extreme care! This is because the shape of the object becomes important in this method: a sharper object will have a higher measured S/N compared to a more diffuse object at the same original magnitude. Besides the inherent shape/sharpness of the object, issues like the PSF also become important in this method (because the finally observed shapes of objects are important here): two surveys with the same surface brightness limit (see Section 7.4.3.6 [Surface brightness limit of image], page 402) will have different magnitude limits if one is taken from space and the other from the ground.

7.4.3.6 Surface brightness limit of image

As we make more observations on one region of the sky and add/combine the observations into one dataset, both the signal and the noise increase. However, the signal increases much faster than the noise: Assuming you add $N$ datasets with equal exposure times, the signal will increases as a multiple of $N$, while noise increases as $\sqrt{N}$. Therefore the signal-to-noise ratio increases by a factor of $\sqrt{N}$. Visually, fainter (per pixel) parts of the objects/signal in the image will become more visible/detectable. The noise-level is known as the dataset’s surface brightness limit.

You can think of the noise as muddy water that is completely covering a flat ground\(^{23}\). The signal (coming from astronomical objects in real data) will be summits/hills that start from the flat sky level (under the muddy water) and their summits can sometimes reach above the muddy water. Let’s assume that in your first observation the muddy water has just been stirred and except a few small peaks, you can’t see anything through the mud. As you wait and make more observations/exposures, the mud settles down and the depth of the transparent water increases. As a result, more and more summits become visible and the lower parts of the hills (parts with lower surface brightness) can be seen more clearly. In this analogy\(^{24}\), height (from the ground) is the surface brightness and the height of the muddy water at the moment you combine your data, is your surface brightness limit for that moment.

---

\(^{23}\) The ground is the sky value in this analogy, see Section 7.1.4 [Sky value], page 344. Note that this analogy only holds for a flat sky value across the surface of the image or ground.

\(^{24}\) Note that this muddy water analogy is not perfect, because while the water-level remains the same all over a peak, in data analysis, the Poisson noise increases with the level of data.
The outputs of NoiseChisel include the Sky standard deviation ($\sigma$) on every group of pixels (a tile) that were calculated from the undetected pixels in each tile, see Section 4.8 [Tessellation], page 189, and Section 7.2.2.3 [NoiseChisel output], page 378. Let’s take $\sigma_m$ as the median $\sigma$ over the successful meshes in the image (prior to interpolation or smoothing). It is recorded in the MEDSTD keyword of the SKY_STD extension of NoiseChisel’s output.

On different instruments, pixels cover different spatial angles over the sky. For example, the width of each pixel on the ACS camera on the Hubble Space Telescope (HST) is roughly 0.05 seconds of arc, while the pixels of SDSS are each 0.396 seconds of arc (almost eight times wider\(^{25}\)). Nevertheless, irrespective of its sky coverage, a pixel is our unit of data collection.

To start with, we define the low-level Surface brightness limit or depth, in units of magnitude/pixel with the equation below (assuming the image has zero point magnitude $z$ and we want the $n$th multiple of $\sigma_m$).

$$SB_{n\sigma,\text{pixel}} = -2.5 \times \log_{10}(n\sigma_m) + z \quad [\text{mag/pixel}]$$

As an example, the XDF survey covers part of the sky that the HST has observed the most (for 85 orbits) and is consequently very small (~4 minutes of arc, squared). On the other hand, the CANDELS survey, is one of the widest multi-color surveys done by the HST covering several fields (about 720 arcmin\(^2\)) but its deepest fields have only 9 orbits observation. The 1$\sigma$ depth of the XDF and CANDELS-deep surveys in the near infrared WFC3/F160W filter are respectively 34.40 and 32.45 magnitudes/pixel. In a single orbit image, this same field has a 1$\sigma$ depth of 31.32 magnitudes/pixel. Recall that a larger magnitude corresponds to less brightness, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

The low-level magnitude/pixel measurement above is only useful when all the datasets you want to use, or compare, have the same pixel size. However, you will often find yourself using, or comparing, datasets from various instruments with different pixel scales (projected pixel width, in arc-seconds). If we know the pixel scale, we can obtain a more easily comparable surface brightness limit in units of: magnitude/arcsec\(^2\). But another complication is that astronomical objects are usually larger than 1 arcsec\(^2\). As a result, it is common to measure the surface brightness limit over a larger (but fixed, depending on context) area.

Let’s assume that every pixel is $p$ arcsec\(^2\) and we want the surface brightness limit for an object covering $A$ arcsec\(^2\) (so $A/p$ is the number of pixels that cover an area of $A$ arcsec\(^2\)). On the other hand, noise is added in RMS\(^{26}\), hence the noise level in $A$ arcsec\(^2\) is $n\sigma_m\sqrt{A/p}$. But we want the result in units of arcsec\(^2\), so we should divide this by $A$ arcsec\(^2\): $n\sigma_m\sqrt{A/p}/A = n\sigma_m\sqrt{A/(pA^2)} = n\sigma_m/\sqrt{pA}$. Plugging this into the magnitude equation, we get the $n\sigma$ surface brightness limit, over an area of $A$ arcsec\(^2\), in units of magnitudes/arcsec\(^2\):

\(^{25}\) Ground-based instruments like the SDSS suffer from strong smoothing due to the atmosphere. Therefore, increasing the pixel resolution (or decreasing the width of a pixel) won’t increase the received information.

\(^{26}\) If you add three datasets with noise $\sigma_1$, $\sigma_2$ and $\sigma_3$, the resulting noise level is $\sigma_t = \sqrt{\sigma_1^2 + \sigma_2^2 + \sigma_3^2}$, so when $\sigma_1 = \sigma_2 = \sigma_3 \equiv \sigma$, then $\sigma_t = \sigma\sqrt{3}$. In this case, the area $A$ is covered by $A/p$ pixels, so the noise level is $\sigma_t = \sigma\sqrt{A/p}$.
\[ SB_{n\sigma,A_{\text{arcsec}^2}} = -2.5 \times \log_{10} \left( \frac{n\sigma_m}{\sqrt{pA}} \right) + z \quad [\text{mag/arcsec}^2] \]

MakeCatalog will calculate the input dataset’s \( SB_{n\sigma,\text{pixel}} \) and \( SB_{n\sigma,A_{\text{arcsec}^2}} \) and will write them as the \texttt{SBLMAGPIX} and \texttt{SBLMAG} keywords the output catalog(s), see Section 7.4.6.4 [MakeCatalog output], page 427. You can set your desired \( n \)-th multiple of \( \sigma \) and the \( A \) \( \text{arcsec}^2 \) area using the following two options respectively: \texttt{--sfmagnsigma} and \texttt{--sfmagarea} (see Section 7.4.6.4 [MakeCatalog output], page 427). Just note that \( SB_{n\sigma,A_{\text{arcsec}^2}} \) is only calculated if the input has World Coordinate System (WCS). Without WCS, the pixel scale cannot be derived.

As you saw in its derivation, the calculation above extrapolates the noise in one pixel over all the input’s pixels! It therefore implicitly assumes that the noise is the same in all of the pixels. But this only happens in individual exposures: reduced data will have correlated noise because they are a stack of many individual exposures that have been warped (thus mixing the pixel values). A more accurate measure which will provide a realistic value for every labeled region is known as the upper-limit magnitude, which is discussed below.

### 7.4.3.7 Upper limit magnitude of image

As mentioned in Section 7.4.3.4 [Upper limit magnitude of each detection], page 400, the upper-limit magnitude will depend on the shape of each object’s footprint. Therefore we can measure a dataset’s upper-limit magnitude using standard shapes.

Traditionally a circular aperture of a fixed size (in arcseconds) has been used. For a full example of implementing this, see the respective section in the tutorial (Section 2.3.4 [Image surface brightness limit], page 88).

### 7.4.4 Measuring elliptical parameters

The shape or morphology of a target is one of the most commonly desired parameters of a target. Here, we will review the derivation of the most basic/simple morphological parameters: the elliptical parameters for a set of labeled pixels. The elliptical parameters are: the (semi-)major axis, the (semi-)minor axis and the position angle along with the central position of the profile. The derivations below follow the SExtractor manual derivations with some added explanations for easier reading.

Let’s begin with one dimension for simplicity: Assume we have a set of \( N \) values \( B_i \) (for example showing the spatial distribution of a target’s brightness), each at position \( x_i \). The simplest parameter we can define is the geometric center of the object \( (x_g) \) (ignoring the brightness values): \( x_g = \frac{\sum_i x_i}{N} \). Moments are defined to incorporate both the value (brightness) and position of the data. The first moment can be written as:

\[ \overline{x} = \frac{\sum_i B_i x_i}{\sum_i B_i} \]

This is essentially the weighted (by \( B_i \)) mean position. The geometric center \( (x_g, \text{defined above}) \) is a special case of this with all \( B_i = 1 \). The second moment is essentially the variance of the distribution:
\[
\bar{x}^2 = \frac{\sum_i B_i(x_i - \bar{x})^2}{\sum_i B_i} = \frac{\sum_i B_i x_i^2}{\sum_i B_i} - 2\bar{x}\frac{\sum_i B_i x_i}{\sum_i B_i} + \bar{x}^2 = \frac{\sum_i B_i x_i^2}{\sum_i B_i} - \bar{x}^2
\]

The last step was done from the definition of \( \bar{x} \). Hence, the square root of \( \bar{x}^2 \) is the spatial standard deviation (along the one-dimension) of this particular brightness distribution \( (B_i) \).

Crudely (or qualitatively), you can think of its square root as the distance (from \( \bar{x} \)) which contains a specific amount of the flux (depending on the \( B_i \) distribution). Similar to the first moment, the geometric second moment can be found by setting all \( B_i = 1 \). So while the first moment quantified the position of the brightness distribution, the second moment quantifies how that brightness is dispersed about the first moment. In other words, it quantifies how “sharp” the object’s image is.

Before continuing to two dimensions and the derivation of the elliptical parameters, let’s pause for an important implementation technicality. You can ignore this paragraph and the next two if you don’t want to implement these concepts. The basic definition (first definition of \( \bar{x} \) above) can be used without any major problem. However, using this fraction requires two runs over the data: one run to find \( \bar{x} \) and another run to find \( \bar{x}^2 \) from \( \bar{x} \), this can be slow. The advantage of the last fraction above, is that we can estimate both the first and second moments in one run (since the \(-\bar{x}^2\) term can easily be added later).

The logarithmic nature of floating point number digitization creates a complication however: suppose the object is located between pixels 10000 and 10020. Hence the target’s pixels are only distributed over 20 pixels (with a standard deviation < 20), while the mean has a value of \( \sim 10000 \). The \( \sum_i B_i^2 x_i^2 \) will go to very very large values while the individual pixel differences will be orders of magnitude smaller. This will lower the accuracy of our calculation due to the limited accuracy of floating point operations. The variance only depends on the distance of each point from the mean, so we can shift all position by a constant/arbitrary \( K \) which is much closer to the mean: \( x - K = \bar{x} - K \). Hence we can calculate the second order moment using:

\[
\bar{x}^2 = \frac{\sum_i B_i(x_i - K)^2}{\sum_i B_i} - (\bar{x} - K)^2
\]

The closer \( K \) is to \( \bar{x} \), the better (the sums of squares will involve smaller numbers), as long as \( K \) is within the object limits (in the example above: \( 10000 \leq K \leq 10020 \)), the floating point error induced in our calculation will be negligible. For the most simplest implementation, MakeCatalog takes \( K \) to be the smallest position of the object in each dimension. Since \( K \) is arbitrary and an implementation/technical detail, we will ignore it for the remainder of this discussion.

In two dimensions, the mean and variances can be written as:

\[
\bar{x} = \frac{\sum_i B_i x_i}{B_i}, \quad \bar{y} = \frac{\sum_i B_i y_i}{B_i}
\]

\[
\bar{x}^2 = \frac{\sum_i B_i x_i^2}{\sum_i B_i} - \bar{x}^2 \quad \bar{y}^2 = \frac{\sum_i B_i y_i^2}{\sum_i B_i} - \bar{y}^2
\]
\[
\bar{xy} = \frac{\sum_i B_i x_i y_i}{\sum_i B_i} - \bar{x} \times \bar{y}
\]

If an elliptical profile’s major axis exactly lies along the \(x\) axis, then \(\bar{x}^2\) will be directly proportional with the profile’s major axis, \(\bar{y}^2\) with its minor axis and \(\bar{xy} = 0\). However, in reality we are not that lucky and (assuming galaxies can be parameterized as an ellipse) the major axis of galaxies can be in any direction on the image (in fact this is one of the core principles behind weak-lensing by shear estimation). So the purpose of the remainder of this section is to define a strategy to measure the position angle and axis ratio of some randomly positioned ellipses in an image, using the raw second moments that we have calculated above in our image coordinates.

Let’s assume we have rotated the galaxy by \(\theta\), the new second order moments are:

\[
\begin{align*}
\bar{x}_\theta &= \bar{x}^2 \cos^2 \theta + \bar{y}^2 \sin^2 \theta - 2\bar{xy} \cos \theta \sin \theta \\
\bar{y}_\theta &= \bar{x}^2 \sin^2 \theta + \bar{y}^2 \cos^2 \theta + 2\bar{xy} \cos \theta \sin \theta \\
\bar{xy}_\theta &= \bar{x}^2 \cos \theta \sin \theta - \bar{y}^2 \cos \theta \sin \theta + \bar{xy}(\cos^2 \theta - \sin^2 \theta)
\end{align*}
\]

The best \(\theta\) (\(\theta_0\), where major axis lies along the \(x_\theta\) axis) can be found by:

\[
\left.\frac{\partial x^2_\theta}{\partial \theta}\right|_{\theta_0} = 0
\]

Taking the derivative, we get:

\[
2 \cos \theta_0 \sin \theta_0 (\bar{y}^2 - \bar{x}^2) + 2(\cos^2 \theta_0 - \sin^2 \theta_0)\bar{xy} = 0
\]

When \(\bar{x}^2 \neq \bar{y}^2\), we can write:

\[
\tan 2\theta_0 = 2 \frac{\bar{xy}}{x^2 - y^2}.
\]

MakeCatalog uses the standard C math library’s \texttt{atan2} function to estimate \(\theta_0\), which we define as the position angle of the ellipse. To recall, this is the angle of the major axis of the ellipse with the \(x\) axis. By definition, when the elliptical profile is rotated by \(\theta_0\), then \(\bar{xy}_{\theta_0} = 0\), \(\bar{x}_{\theta_0}^2\) will be the extent of the maximum variance and \(\bar{y}_{\theta_0}^2\) the extent of the minimum variance (which are perpendicular for an ellipse). Replacing \(\theta_0\) in the equations above for \(\bar{x}_\theta\) and \(\bar{y}_\theta\), we can get the semi-major (\(A\)) and semi-minor (\(B\)) lengths:

\[
A^2 \equiv \frac{\bar{x}_{\theta_0}^2}{2} = \frac{\bar{x}^2 + \bar{y}^2}{2} + \sqrt{\left(\frac{\bar{x}^2 - \bar{y}^2}{2}\right)^2 + \bar{xy}^2}
\]
\[ B^2 = \frac{y_0^2}{2} \equiv \frac{x^2 + y^2}{2} - \sqrt{\left(\frac{x^2 - y^2}{2}\right)^2 + \pi y^2} \]

As a summary, it is important to remember that the units of \( A \) and \( B \) are in pixels (the standard deviation of a positional distribution) and that they represent the spatial light distribution of the object in both image dimensions (rotated by \( \theta_0 \)). When the object cannot be represented as an ellipse, this interpretation breaks down: \( \pi y_0^2 \neq 0 \) and \( y_0^2 \) will not be the direction of minimum variance.

7.4.5 Adding new columns to MakeCatalog

MakeCatalog is designed to allow easy addition of different measurements over a labeled image (see Akhlaghi [2016] (https://arxiv.org/abs/1611.06387v1)). A check-list style description of necessary steps to do that is described in this section. The common development characteristics of MakeCatalog and other Gnuastro programs is explained in Chapter 12 [Developing], page 670. We strongly encourage you to have a look at that chapter to greatly simplify your navigation in the code. After adding and testing your column, you are most welcome (and encouraged) to share it with us so we can add to the next release of Gnuastro for everyone else to also benefit from your efforts.

MakeCatalog will first pass over each label’s pixels two times and do necessary raw/internal calculations. Once the passes are done, it will use the raw information for filling the final catalog’s columns. In the first pass it will gather mainly object information and in the second run, it will mainly focus on the clumps, or any other measurement that needs an output from the first pass. These two passes are designed to be raw summations: no extra processing. This will allow parallel processing and simplicity/clarity. So if your new calculation, needs new raw information from the pixels, then you will need to also modify the respective \texttt{mkcatalog\_first\_pass} and \texttt{mkcatalog\_second\_pass} functions (both in \texttt{bin/mkcatalog/mkcatalog.c}) and define new raw table columns in \texttt{main.h} (hopefully the comments in the code are clear enough).

In all these different places, the final columns are sorted in the same order (same order as Section 7.4.6 [Invoking MakeCatalog], page 408). This allows a particular column/option to be easily found in all steps. Therefore in adding your new option, be sure to keep it in the same relative place in the list in all the separate places (it doesn’t necessarily have to be in the end), and near conceptually similar options.

\texttt{main.h}  

The \texttt{objectcols} and \texttt{clumpcols} enumerated variables (\texttt{enum}) define the raw/internal calculation columns. If your new column requires new raw calculations, add a row to the respective list. If your calculation requires any other settings parameters, you should add a variable to the \texttt{mkcatalogparams} structure.

\texttt{ui.c}  

If the new column needs raw calculations (an entry was added in \texttt{objectcols} and \texttt{clumpcols}), specify which inputs it needs in \texttt{ui\_necessary\_inputs}, similar to the other options. Afterwards, if your column includes any particular settings (you needed to add a variable to the \texttt{mkcatalogparams} structure in \texttt{main.h}), you should do the sanity checks and preparations for it here.
ui.h  The option_keys_enum associates a unique value for each option to MakeCatalog. The options that have a short option version, the single character short comment is used for the value. Those that don’t have a short option version, get a large integer automatically. You should add a variable here to identify your desired column.

args.h  This file specifies all the parameters for the GNU C library, Argp structure that is in charge of reading the user’s options. To define your new column, just copy an existing set of parameters and change the first, second and 5th values (the only ones that differ between all the columns), you should use the macro you defined in ui.h here.

columns.c  This file contains the main definition and high-level calculation of your new column through the columns_define_alloc and columns_fill functions. In the first, you specify the basic information about the column: its name, units, comments, type (see Section 4.5 [Numeric data types], page 178) and how it should be printed if the output is a text file. You should also specify the raw/internal columns that are necessary for this column here as the many existing examples show. Through the types for objects and rows, you can specify if this column is only for clumps, objects or both.

The second main function (columns_fill) writes the final value into the appropriate column for each object and clump. As you can see in the many existing examples, you can define your processing on the raw/internal calculations here and save them in the output.

mkcatalog.c  This file contains the low-level parsing functions. To be optimized, the parsing is done in parallel through the mkcatalog_single_object function. This function initializes the necessary arrays and calls the lower-level parse_objects and parse_clumps for actually going over the pixels. They are all heavily commented, so you should be able to follow where to add your necessary low-level calculations.

doc/gnuastro.texi  Update this manual and add a description for the new column.

7.4.6 Invoking MakeCatalog

MakeCatalog will do measurements and produce a catalog from a labeled dataset and optional values dataset(s). The executable name is astmkcatalog with the following general template

$ astmkcatalog [OPTION ...] InputImage.fits

One line examples:

## Create catalog with RA, Dec, Magnitude and Magnitude error,  
## from Segment’s output:
$ astmkcatalog --ra --dec --magnitude --magnitudeerr seg-out.fits

## Same catalog as above (using short options):

$ asmkcatalog -rdmG seg-out.fits

## Write the catalog to a text table:
$ asmkcatalog -mpQ seg-out.fits --output=cat.txt

## Output columns specified in ‘columns.conf’:
$ asmkcatalog --config=columns.conf seg-out.fits

## Use object and clump labels from a K-band image, but pixel values ## from an i-band image.
$ asmkcatalog K_segmented.fits --hdu=DETECTIONS --clumpscat \ 
   --clumpsfile=K_segmented.fits --clumpshdu=CLUMPS \ 
   --valuesfile=i_band.fits

If MakeCatalog is to do processing (not printing help or option values), an input labeled image should be provided. The options described in this section are those that are particular to MakeProfiles. For operations that MakeProfiles shares with other programs (mainly involving input/output or general processing steps), see Section 4.1.2 [Common options], page 156. Also see Chapter 4 [Common program behavior], page 152, for some general characteristics of all Gnuastro programs including MakeCatalog.

The various measurements/columns of MakeCatalog are requested as options, either on the command-line or in configuration files, see Section 4.2 [Configuration files], page 169. The full list of available columns is available in Section 7.4.6.3 [MakeCatalog measurements], page 415. Depending on the requested columns, MakeCatalog needs more than one input dataset, for more details, please see Section 7.4.6.1 [MakeCatalog inputs and basic settings], page 409. The upper-limit measurements in particular need several configuration options which are thoroughly discussed in Section 7.4.6.2 [Upper-limit settings], page 413. Finally, in Section 7.4.6.4 [MakeCatalog output], page 427, the output file(s) created by MakeCatalog are discussed.

7.4.6.1 MakeCatalog inputs and basic settings

MakeCatalog works by using a localized/labeled dataset (see Section 7.4 [MakeCatalog], page 392). This dataset maps/labels pixels to a specific target (row number in the final catalog) and is thus the only necessary input dataset to produce a minimal catalog in any situation. Because it only has labels/counters, it must have an integer type (see Section 4.5 [Numeric data types], page 178), see below if your labels are in a floating point container. When the requested measurements only need this dataset (for example --geox, --geoy, or --geoarea), MakeCatalog won’t read any more datasets.

Low-level measurements that only use the labeled image are rarely sufficient for any high-level science case. Therefore necessary input datasets depend on the requested columns in each run. For example, let’s assume you want the brightness/magnitude and signal-to-noise ratio of your labeled regions. For these columns, you will also need to provide an extra dataset containing values for every pixel of the labeled input (to measure brightness) and another for the Sky standard deviation (to measure error). All such auxiliary input files have to have the same size (number of pixels in each dimension) as the input labeled image. Their numeric data type is irrelevant (they will be converted to 32-bit floating point
internally). For the full list of available measurements, see Section 7.4.6.3 [MakeCatalog measurements], page 415.

The “values” dataset is used for measurements like brightness/magnitude, or flux-weighted positions. If it is a real image, by default it is assumed to be already Sky-subtracted prior to running MakeCatalog. If it isn’t, you use the --subtractsky option to, so MakeCatalog reads and subtracts the Sky dataset before any processing. To obtain the Sky value, you can use the --sky option of Section 7.1 [Statistics], page 337, but the best recommended method is Section 7.2 [NoiseChisel], page 362, see Section 7.1.4 [Sky value], page 344.

MakeCatalog can also do measurements on sub-structures of detections. In other words, it can produce two catalogs. Following the nomenclature of Segment (see Section 7.3 [Segment], page 381), the main labeled input dataset is known as “object” labels and the (optional) sub-structure input dataset is known as “clumps”. If MakeCatalog is run with the --clumpscat option, it will also need a labeled image containing clumps, similar to what Segment produces (see Section 7.3.1.3 [Segment output], page 390). Since clumps are defined within detected regions (they exist over signal, not noise), MakeCatalog uses their boundaries to subtract the level of signal under them.

There are separate options to explicitly request a file name and HDU/extension for each of the required input datasets as fully described below (with the --*file format). When each dataset is in a separate file, these options are necessary. However, one great advantage of the FITS file format (that is heavily used in astronomy) is that it allows the storage of multiple datasets in one file. So in most situations (for example if you are using the outputs of Section 7.2 [NoiseChisel], page 362, or Section 7.3 [Segment], page 381), all the necessary input datasets can be in one file.

When none of the --*file options are given, MakeCatalog will assume the necessary input datasets are in the file given as its argument (without any option). When the Sky or Sky standard deviation datasets are necessary and the only --*file option called is --valuesfile, MakeCatalog will search for these datasets (with the default/given HDUs) in the file given to --valuesfile (before looking into the main argument file).

When the clumps image (necessary with the --clumpscat option) is used, MakeCatalog looks into the (possibly existing) NUMLABS keyword for the total number of clumps in the image (irrespective of how many objects there are). If its not present, it will count them and possibly re-label the clumps so the clump labels always start with 1 and finish with the total number of clumps in each object. The re-labeled clumps image will be stored with the -clumps-relab.fits suffix. This can slightly slow-down the run.

Note that NUMLABS is automatically written by Segment in its outputs, so if you are feeding Segment’s clump labels, you can benefit from the improved speed. Otherwise, if you are creating the clumps label dataset manually, it may be good to include the NUMLABS keyword in its header and also be sure that there is no gap in the clump labels. For example if an object has three clumps, they are labeled as 1, 2, 3. If they are labeled as 1, 3, 4, or any other combination of three positive integers that aren’t an increment of the previous, you might get unknown behavior.

It may happen that your labeled objects image was created with a program that only outputs floating point files. However, you know it only has integer valued pixels that are stored in a floating point container. In such cases, you can use Gnuastro’s Arithmetic
program (see Section 6.2 [Arithmetic], page 272) to change the numerical data type of the image (float.fits) to an integer type image (int.fits) with a command like below:

$ astarithmetic float.fits int32 --output=int.fits

To summarize: if the input file to MakeCatalog is the default/full output of Segment (see Section 7.3.1.3 [Segment output], page 390) you don’t have to worry about any of the --*file options below. You can just give Segment’s output file to MakeCatalog as described in Section 7.4.6 [Invoking MakeCatalog], page 408. To feed NoiseChisel’s output into MakeCatalog, just change the labeled dataset’s header (with --hdu=DETECTIONS). The full list of input dataset options and general setting options are described below.

-1 FITS
--clumpsfile=FITS

The FITS file containing the labeled clumps dataset when --clumpscat is called (see Section 7.4.6.4 [MakeCatalog output], page 427). When --clumpscat is called, but this option isn’t, MakeCatalog will look into the main input file (given as an argument) for the required extension/HDU (value to --clumpshdu).

--clumpshdu=STR

The HDU/extension of the clump labels dataset. Only pixels with values above zero will be considered. The clump labels dataset has to be an integer data type (see Section 4.5 [Numeric data types], page 178) and only pixels with a value larger than zero will be used. See Section 7.3.1.3 [Segment output], page 390, for a description of the expected format.

-v FITS
--valuesfile=FITS

The file name of the (sky-subtracted) values dataset. When any of the columns need values to associate with the input labels (for example to measure the brightness/magnitude of a galaxy), MakeCatalog will look into a “values” for the respective pixel values. In most common processing, this is the actual astronomical image that the labels were defined, or detected, over. The HDU/extension of this dataset in the given file can be specified with --valueshdu. If this option is not called, MakeCatalog will look for the given extension in the main input file.

--valueshdu=STR/INT

The name or number (counting from zero) of the extension containing the “values” dataset, see the descriptions above and those in --valuesfile for more.

-s FITS/FLT
--insky=FITS/FLT

Sky value as a single number, or the file name containing a dataset (different values per pixel or tile). The Sky dataset is only necessary when --subtractsky is called or when a column directly related to the Sky value is requested (currently --sky). This dataset may be a tessellation, with one element per tile (see --onelempertile of NoiseChisel’s Section 4.1.2.2 [Processing options], page 159).

When the Sky dataset is necessary but this option is not called, MakeCatalog will assume it is an HDU/extension (specified by --skyhdu) in one of the already
given files. First it will look for it in the --valuesfile (if it is given) and then
the main input file (given as an argument).
By default the values dataset is assumed to be already Sky subtracted, so this
dataset is not necessary for many of the columns.

--skyhdu=STR
HDU/extension of the Sky dataset, see --skyfile.

--subtractsky
Subtract the sky value or dataset from the values file prior to any processing.

-t STR/FLT
--instd=STR/FLT
Sky standard deviation value as a single number, or the file name containing
a dataset (different values per pixel or tile). With the --variance option you
can tell MakeCatalog to interpret this value/dataset as a variance image, not
standard deviation.
Important note: This must only be the SKY standard deviation or variance
(not including the signal’s contribution to the error). In other words, the fi-
nal standard deviation of a pixel depends on how much signal there is in it.
MakeCatalog will find the amount of signal within each pixel (while subtract-
ing the Sky, if --subtractsky is called) and account for the extra error due
to it’s value (signal). Therefore if the input standard deviation (or variance)
image also contains the contribution of signal to the error, then the final error
measurements will be over-estimated.

--stdhdu=STR
The HDU of the Sky value standard deviation image.

--variance
The dataset given to --instd (and --stdhdu has the Sky variance of every
pixel, not the Sky standard deviation.

--forcereadstd
Read the input STD image even if it is not required by any of the requested
columns. This is because some of the output catalog’s metadata may need it,
for example to calculate the dataset’s surface brightness limit (see Section 7.4.3
[Quantifying measurement limits], page 398, configured with --sfmagarea and
--sfmagnsigma in Section 7.4.6.4 [MakeCatalog output], page 427).
Furthermore, if the input STD image doesn’t have the MEDSTD keyword (that is
meant to contain the representative standard deviation of the full image), with
this option, the median will be calculated and used for the surface brightness
limit.

-z FLT
--zeropoint=FLT
The zero point magnitude for the input image, see Section 7.4.2 [Brightness,
Flux, Magnitude and Surface brightness], page 394.

--sigmaclip FLT,FLT
The sigma-clipping parameters when any of the sigma-clipping related columns
are requested (for example --sigclip-median or --sigclip-number).
This option takes two values: the first is the multiple of $\sigma$, and the second is the termination criteria. If the latter is larger than 1, it is read as an integer number and will be the number of times to clip. If it is smaller than 1, it is interpreted as the tolerance level to stop clipping. See Section 7.1.3 [Sigma clipping], page 343, for a complete explanation.

`--fracmax=FLT[,FLT]`

The fractions (one or two) of maximum value in objects or clumps to be used in the related columns, for example `--fracmaxarea1`, `--fracmaxsum1` or `--fracmaxradius1`, see Section 7.4.6.3 [MakeCatalog measurements], page 415. For the maximum value, see the description of `--maximum` column below. The value(s) of this option must be larger than 0 and smaller than 1 (they are a fraction). When only `--fracmaxarea1` or `--fracmaxsum1` is requested, one value must be given to this option, but if `--fracmaxarea2` or `--fracmaxsum2` are also requested, two values must be given to this option. The values can be written as simple floating point numbers, or as fractions, for example 0.25, 0.75 and 0.25, $\frac{3}{4}$ are the same.

`--spatialresolution=FLT`

The error in measuring spatial properties (for example the area) in units of pixels. You can think of this as the FWHM of the dataset’s PSF and is used in measurements like the error in surface brightness (`--sberror`, see Section 7.4.6.3 [MakeCatalog measurements], page 415). Ideally, images are taken in the optimal Nyquist sampling Section 6.3.2.7 [Sampling theorem], page 318, so the default value for this option is 2. But in practice real images my be oversampled (usually ground-based images, where you will need to increase the default value) or undersampled (some space-based images, where you will need to decrease the default value).

### 7.4.6.2 Upper-limit settings

The upper-limit magnitude was discussed in Section 7.4.3 [Quantifying measurement limits], page 398. Unlike other measured values/columns in MakeCatalog, the upper limit magnitude needs several extra parameters which are discussed here. All the options specific to the upper-limit measurements start with `up` for “upper-limit”. The only exception is `--envseed` that is also present in other programs and is general for any job requiring random number generation in Gnuastro (see Section 8.2.1.4 [Generating random numbers], page 464).

One very important consideration in Gnuastro is reproducibility. Therefore, the values to all of these parameters along with others (like the random number generator type and seed) are also reported in the comments of the final catalog when the upper limit magnitude column is desired. The random seed that is used to define the random positions for each object or clump is unique and set based on the (optionally) given seed, the total number of objects and clumps and also the labels of the clumps and objects. So with identical inputs, an identical upper-limit magnitude will be found. However, even if the seed is identical, when the ordering of the object/clump labels differs between different runs, the result of upper-limit measurements will not be identical.

MakeCatalog will randomly place the object/clump footprint over the dataset. When the randomly placed footprint doesn’t fall on any object or masked region (see `--upmaskfile`)
it will be used in the final distribution. Otherwise that particular random position will be ignored and another random position will be generated. Finally, when the distribution has the desired number of successfully measured random samples (--upnum) the distribution's properties will be measured and placed in the catalog.

When the profile is very large or the image is significantly covered by detections, it might not be possible to find the desired number of samplings in a reasonable time. MakeProfiles will continue searching until it is unable to find a successful position (since the last successful measurement\(^{27}\)), for a large multiple of --upnum (currently\(^{28}\) this is 10). If --upnum successful samples cannot be found until this limit is reached, MakeCatalog will set the upper-limit magnitude for that object to NaN (blank).

MakeCatalog will also print a warning if the range of positions available for the labeled region is smaller than double the size of the region. In such cases, the limited range of random positions can artificially decrease the standard deviation of the final distribution. If your dataset can allow it (it is large enough), it is recommended to use a larger range if you see such warnings.

```plaintext
--upmaskfile=FITS
```

File name of mask image to use for upper-limit calculation. In some cases (especially when doing matched photometry), the object labels specified in the main input and mask image might not be adequate. In other words they do not necessarily have to cover all detected objects: the user might have selected only a few of the objects in their labeled image. This option can be used to ignore regions in the image in these situations when estimating the upper-limit magnitude. All the non-zero pixels of the image specified by this option (in the --upmaskhdu extension) will be ignored in the upper-limit magnitude measurements.

For example, when you are using labels from another image, you can give NoiseChisel's objects image output for this image as the value to this option. In this way, you can be sure that regions with data do not harm your distribution. See Section 7.4.3 [Quantifying measurement limits], page 398, for more on the upper limit magnitude.

```plaintext
--upmaskhdu=STR
```

The extension in the file specified by --upmask.

```plaintext
--upnum=INT
```

The number of random samples to take for all the objects. A larger value to this option will give a more accurate result (asymptotically), but it will also slow down the process. When a randomly positioned sample overlaps with a detected/masked pixel it is not counted and another random position is found until the object completely lies over an undetected region. So you can be sure that for each object, this many samples over undetected objects are made. See the upper limit magnitude discussion in Section 7.4.3 [Quantifying measurement limits], page 398, for more.

\(^{27}\) The counting of failed positions restarts on every successful measurement.

\(^{28}\) In Gnuastro's source, this constant number is defined as the MKCATALOG_UPPERLIMIT_MAXFAILS_MULTIP macro in bin/mkcatalog/main.h, see Section 3.2 [Downloading the source], page 131.
Chapter 7: Data analysis

--uprange=INT,INT
The range/width of the region (in pixels) to do random sampling along each dimension of the input image around each object’s position. This is not a mandatory option and if not given (or given a value of zero in a dimension), the full possible range of the dataset along that dimension will be used. This is useful when the noise properties of the dataset vary gradually. In such cases, using the full range of the input dataset is going to bias the result. However, note that decreasing the range of available positions too much will also artificially decrease the standard deviation of the final distribution (and thus bias the upper-limit measurement).

--envseed
Read the random number generator type and seed value from the environment (see Section 8.2.1.4 [Generating random numbers], page 464). Random numbers are used in calculating the random positions of different samples of each object.

--upsigmaclip=FLT,FLT
The raw distribution of random values will not be used to find the upper-limit magnitude, it will first be \( \sigma \)-clipped (see Section 7.1.3 [Sigma clipping], page 343) to avoid outliers in the distribution (mainly the faint undetected wings of bright/large objects in the image). This option takes two values: the first is the multiple of \( \sigma \), and the second is the termination criteria. If the latter is larger than 1, it is read as an integer number and will be the number of times to clip. If it is smaller than 1, it is interpreted as the tolerance level to stop clipping. See Section 7.1.3 [Sigma clipping], page 343, for a complete explanation.

--upnsigma=FLT
The multiple of the final (\( \sigma \)-clipped) standard deviation (or \( \sigma \)) used to measure the upper-limit brightness or magnitude.

--checkuplim=INT[,INT]
Print a table of positions and measured values for all the full random distribution used for one particular object or clump. If only one integer is given to this option, it is interpreted to be an object’s label. If two values are given, the first is the object label and the second is the ID of requested clump within it.
The output is a table with three columns (its type is determined with the --tableformat option, see Section 4.1.2.1 [Input/Output options], page 157). The first two columns are the position of the first pixel in each random sampling of this particular object/clump. The third column is the measured flux over that region. If the region overlapped with a detection or masked pixel, then its measured value will be a NaN (not-a-number). The total number of rows is thus unknown, but you can be sure that the number of rows with non-NaN measurements is the number given to the --upnum option.

7.4.6.3 MakeCatalog measurements
The final group of options particular to MakeCatalog are those that specify which measurements/columns should be written into the final output table. The current measurements in MakeCatalog are those which only produce one final value for each label (for example
its total brightness: a single number). All the different label’s measurements can be written as one column in a final table/catalog that contains other columns for other similar single-number measurements.

In this case, all the different label’s measurements can be written as one column in a final table/catalog that contains other columns for other similar single-number measurements. The majority of this section is devoted to MakeCatalog’s single-valued measurements. However, MakeCatalog can also do measurements that produce more than one value for each label. Currently the only such measurement is generation of spectra from 3D cubes with the `--spectrum` option and it is discussed in the end of this section.

Command-line options are used to identify which measurements you want in the final catalog(s) and in what order. If any of the options below is called on the command line or in any of the configuration files, it will be included as a column in the output catalog. The order of the columns is in the same order as the options were seen by MakeCatalog (see Section 4.2.2 [Configuration file precedence], page 170). Some of the columns apply to both “objects” and “clumps” and some are particular to only one of them (for the definition of “objects” and “clumps”, see Section 7.3 [Segment], page 381). Columns/options that are unique to one catalog (only objects, or only clumps), are explicitly marked with [Objects] or [Clumps] to specify the catalog they will be placed in.

**--i**
This is a unique option which can add multiple columns to the final catalog(s). Calling this option will put the object IDs (`--objid`) in the objects catalog and host-object-ID (`--hostobjid`) and ID-in-host-object (`--idinhostobj`) into the clumps catalog. Hence if only object catalogs are required, it has the same effect as `--objid`.

**--objid** [Objects] ID of this object.

**--hostobjid** [Clumps] The ID of the object which hosts this clump.

**--idinhostobj** [Clumps] The ID of this clump in its host object.

**--x**
The flux weighted center of all objects and clumps along the first FITS axis (horizontal when viewed in SAO DS9), see **x** in Section 7.4.4 [Measuring elliptical parameters], page 404. The weight has to have a positive value (pixel value larger than the Sky value) to be meaningful! Specially when doing matched photometry, this might not happen: no pixel value might be above the Sky value. For such detections, the geometric center will be reported in this column (see `--geox`). You can use `--weightarea` to see which was used.

**--y**
The flux weighted center of all objects and clumps along the second FITS axis (vertical when viewed in SAO DS9). See `--x`.

**--z**
The flux weighted center of all objects and clumps along the third FITS axis. See `--x`. 
--geox  The geometric center of all objects and clumps along the first FITS axis axis. The geometric center is the average pixel positions irrespective of their pixel values.

--geoy  The geometric center of all objects and clumps along the second FITS axis axis,

--geoz  The geometric center of all objects and clumps along the third FITS axis axis,

--minvx  Position of pixel with minimum value in objects and clumps, along the first

--maxvx  Position of pixel with minimum value in objects and clumps, along the first

--minvy  Position of pixel with minimum value in objects and clumps, along the first

--maxvy  Position of pixel with minimum value in objects and clumps, along the first

--minvz  Position of pixel with minimum value in objects and clumps, along the first

--maxvz  Position of pixel with minimum value in objects and clumps, along the first

--minx  The minimum position of all objects and clumps along the first FITS axis.

--maxx  The maximum position of all objects and clumps along the first FITS axis.

--miny  The minimum position of all objects and clumps along the second FITS axis.

--maxy  The maximum position of all objects and clumps along the second FITS axis.

--minz  The minimum position of all objects and clumps along the third FITS axis.

--maxz  The maximum position of all objects and clumps along the third FITS axis.

--clumpsx  [Objects] The flux weighted center of all the clumps in this object along the

--clumpsy  [Objects] The flux weighted center of all the clumps in this object along the

--clumpsz  [Objects] The flux weighted center of all the clumps in this object along the

--clumpsgeox  [Objects] The geometric center of all the clumps in this object along the first

--clumpsgeoy  [Objects] The geometric center of all the clumps in this object along the second

--clumpsgeoz  [Objects] The geometric center of all the clumps in this object along the third


--clumpsgeoz
  [Objects] The geometric center of all the clumps in this object along the third FITS axis. See --geoz.

-r
--ra
  Flux weighted right ascension of all objects or clumps, see --x. This is just an alias for one of the lower-level --w1 or --w2 options. Using the FITS WCS keywords (CTYPE), MakeCatalog will determine which axis corresponds to the right ascension. If no CTYPE keywords start with RA, an error will be printed when requesting this column and MakeCatalog will abort.

-d
--dec
  Flux weighted declination of all objects or clumps, see --x. This is just an alias for one of the lower-level --w1 or --w2 options. Using the FITS WCS keywords (CTYPE), MakeCatalog will determine which axis corresponds to the declination. If no CTYPE keywords start with DEC, an error will be printed when requesting this column and MakeCatalog will abort.

--w1
  Flux weighted first WCS axis of all objects or clumps, see --x. The first WCS axis is commonly used as right ascension in images.

--w2
  Flux weighted second WCS axis of all objects or clumps, see --x. The second WCS axis is commonly used as declination in images.

--w3
  Flux weighted third WCS axis of all objects or clumps, see --x. The third WCS axis is commonly used as wavelength in integral field unit data cubes.

--geow1
  Geometric center in first WCS axis of all objects or clumps, see --geox. The first WCS axis is commonly used as right ascension in images.

--geow2
  Geometric center in second WCS axis of all objects or clumps, see --geox. The second WCS axis is commonly used as declination in images.

--geow3
  Geometric center in third WCS axis of all objects or clumps, see --geox. The third WCS axis is commonly used as wavelength in integral field unit data cubes.

--clumpsw1
  [Objects] Flux weighted center in first WCS axis of all clumps in this object, see --x. The first WCS axis is commonly used as right ascension in images.

--clumpsw2
  [Objects] Flux weighted declination of all clumps in this object, see --x. The second WCS axis is commonly used as declination in images.

--clumpsw3
  [Objects] Flux weighted center in third WCS axis of all clumps in this object, see --x. The third WCS axis is commonly used as wavelength in integral field unit data cubes.

--clumpsgeow1
  [Objects] Geometric center right ascension of all clumps in this object, see --geox. The first WCS axis is commonly used as right ascension in images.
--clumpsgeow2

[Objects] Geometric center declination of all clumps in this object, see --geox. The second WCS axis is commonly used as declination in images.

--clumpsgeow3

[Objects] Geometric center in third WCS axis of all clumps in this object, see --geox. The third WCS axis is commonly used as wavelength in integral field unit data cubes.

--brightness

The brightness (sum of all pixel values), see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394. For clumps, the ambient brightness (flux of river pixels around the clump multiplied by the area of the clump) is removed, see --riverave. So the sum of all the clumps brightness in the clump catalog will be smaller than the total clump brightness in the --clumpbrightness column of the objects catalog.

If no usable pixels are present over the clump or object (for example they are all blank), the returned value will be NaN (note that zero is meaningful).

--brightnesserr

The (1σ) error in measuring the brightness of a label (objects or clumps). The returned value will be NaN when the label covers only NaN pixels in the values image, or a pixel is NaN in the --instd image, but non-NaN in the values image. The latter situation usually happens when there is a bug in the previous steps of your analysis, and is important because those pixels with a NaN in the --instd image may contribute significantly to the final error. If you want to ignore those pixels in the error measurement, set them to zero (which is a meaningful number in such scenarios).

--clumpbrightness

[Objects] The total brightness of the clumps within an object. This is simply the sum of the pixels associated with clumps in the object. If no usable pixels are present over the clump or object (for example they are all blank), the stored value will be NaN (note that zero is meaningful).

--brightnessnoriver

[Clumps] The Sky (not river) subtracted clump brightness. By definition, for the clumps, the average brightness of the rivers surrounding it are subtracted from it for a first order accounting for contamination by neighbors. In cases where you will be calculating the flux brightness difference later (one example below) the contamination will be (mostly) removed at that stage, which is why this column was added.

If no usable pixels are present over the clump or object (for example they are all blank), the stored value will be NaN (note that zero is meaningful).

--mean

The mean sky subtracted value of pixels within the object or clump. For clumps, the average river flux is subtracted from the sky subtracted mean.

--median

The median sky subtracted value of pixels within the object or clump. For clumps, the average river flux is subtracted from the sky subtracted median.
--maximum
The maximum value of pixels within the object or clump. When the label (object or clump) is larger than three pixels, the maximum is actually derived by the mean of the brightest three pixels, not the largest pixel value of the same label. This is because noise fluctuations can be very strong in the extreme values of the objects/clumps due to Poisson noise (which gets stronger as the mean gets higher). Simply using the maximum pixel value will create a strong scatter in results that depend on the maximum (for example the --fwhm option also uses this value internally).

--sigclip-number
The number of elements/pixels in the dataset after sigma-clipping the object or clump. The sigma-clipping parameters can be set with the --sigmaclip option described in Section 7.4.6.1 [MakeCatalog inputs and basic settings], page 409. For more on Sigma-clipping, see Section 7.1.3 [Sigma clipping], page 343.

--sigclip-median
The sigma-clipped median value of the object of clump's pixel distribution. For more on sigma-clipping and how to define it, see --sigclip-number.

--sigclip-mean
The sigma-clipped mean value of the object of clump's pixel distribution. For more on sigma-clipping and how to define it, see --sigclip-number.

--sigclip-std
The sigma-clipped standard deviation of the object of clump's pixel distribution. For more on sigma-clipping and how to define it, see --sigclip-number.

-m
--magnitude
The magnitude of clumps or objects, see --brightness.

-e
--magnitudeerr
The magnitude error of clumps or objects. The magnitude error is calculated from the signal-to-noise ratio (see --sn and Section 7.4.3 [Quantifying measurement limits], page 398). Note that until now this error assumes uncorrelated pixel values and also does not include the error in estimating the aperture (or error in generating the labeled image).

For now these factors have to be found by other means. Task 14124 (https://savannah.gnu.org/task/index.php?14124) has been defined for work on adding these sources of error too.

The returned value will be NaN when the label covers only NaN pixels in the values image, or a pixel is NaN in the --instd image, but non-NaN in the values image. The latter situation usually happens when there is a bug in the previous steps of your analysis, and is important because those pixels with a NaN in the --instd image may contribute significantly to the final error. If you want to ignore those pixels in the error measurement, set them to zero (which is a meaningful number in such scenarios).
--clumpsmagnitude
[Objects] The magnitude of all clumps in this object, see --clumpbrightness.

--upperlimit
The upper limit value (in units of the input image) for this object or clump. This is the sigma-clipped standard deviation of the random distribution, multiplied by the value of --upnsigma. See Section 7.4.3 [Quantifying measurement limits], page 398, and Section 7.4.6.2 [Upper-limit settings], page 413, for a complete explanation. This is very important for the fainter and smaller objects in the image where the measured magnitudes are not reliable.

--upperlimitmag
The upper limit magnitude for this object or clump. See Section 7.4.3 [Quantifying measurement limits], page 398, and Section 7.4.6.2 [Upper-limit settings], page 413, for a complete explanation. This is very important for the fainter and smaller objects in the image where the measured magnitudes are not reliable.

--upperlimitsb
The upper-limit surface brightness (in units of mag/arcsec\(^2\)) of this labeled region (object or clump). This is just a simple wrapper over lower-level columns: setting B and A as the value in the columns --upperlimit and --areaarcsec2, we fill this column by simply use the surface brightness equation of Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

--upperlimitonesigma
The 1\(\sigma\) upper limit value (in units of the input image) for this object or clump. See Section 7.4.3 [Quantifying measurement limits], page 398, and Section 7.4.6.2 [Upper-limit settings], page 413, for a complete explanation. When --upnsigma=1, this column’s values will be the same as --upperlimit.

--upperlimitsigma
The position of the total brightness measured within the distribution of randomly placed upperlimit measurements in units of the distribution’s \(\sigma\) or standard deviation. See Section 7.4.3 [Quantifying measurement limits], page 398, and Section 7.4.6.2 [Upper-limit settings], page 413, for a complete explanation.

--upperlimitquantile
The position of the total brightness measured within the distribution of randomly placed upperlimit measurements as a quantile (value between 0 or 1). See Section 7.4.3 [Quantifying measurement limits], page 398, and Section 7.4.6.2 [Upper-limit settings], page 413, for a complete explanation. If the object is brighter than the brightest randomly placed profile, a value of inf is returned. If it is less than the minimum, a value of -inf is reported.

--upperlimitskew
This column contains the non-parametric skew of the \(\sigma\)-clipped random distribution that was used to estimate the upper-limit magnitude. Taking \(\mu\) as the mean, \(\nu\) as the median and \(\sigma\) as the standard deviation, the traditional definition of skewness is defined as: \((\mu - \nu)/\sigma\).

This can be a good measure to see how much you can trust the random measurements, or in other words, how accurately the regions with signal have been
masked/detected. If the skewness is strong (and to the positive), then you can tell that you have a lot of undetected signal in the dataset, and therefore that the upper-limit measurement (and other measurements) are not reliable.

--riverave
[Clumps] The average brightness of the river pixels around this clump. River pixels were defined in Akhlaghi and Ichikawa 2015. In short they are the pixels immediately outside of the clumps. This value is used internally to find the brightness (or magnitude) and signal to noise ratio of the clumps. It can generally also be used as a scale to gauge the base (ambient) flux surrounding the clump. In case there was no river pixels, then this column will have the value of the Sky under the clump. So note that this value is not sky subtracted.

--rivernum
[Clumps] The number of river pixels around this clump, see --riverave.

-n
--sn
The Signal to noise ratio (S/N) of all clumps or objects. See Akhlaghi and Ichikawa (2015) for the exact equations used.
The returned value will be NaN when the label covers only NaN pixels in the values image, or a pixel is NaN in the --instd image, but non-NaN in the values image. The latter situation usually happens when there is a bug in the previous steps of your analysis, and is important because those pixels with a NaN in the --instd image may contribute significantly to the final error. If you want to ignore those pixels in the error measurement, set them to zero (which is a meaningful number).

--sky
The sky flux (per pixel) value under this object or clump. This is actually the mean value of all the pixels in the sky image that lie on the same position as the object or clump.

--std
The sky value standard deviation (per pixel) for this clump or object. This is the square root of the mean variance under the object, or the root mean square.

-C
--numclumps
[Objects] The number of clumps in this object.

-a
--area
The raw area (number of pixels/voxels) in any clump or object independent of what pixel it lies over (if it is NaN/blank or unused for example).

--areaarcsec2
The used (non-blank in values image) area of the labeled region in units of arc-seconds squared. This column is just the value of the --area column, multiplied by the area of each pixel in the input image (in units of arcsec^2). Similar to the --ra or --dec columns, for this option to work, the objects extension used has to have a WCS structure.

--areaminv
The number of pixels that are equal to the minimum value of the labeled region (clump or object).
--areamaxv
The number of pixels that are equal to the maximum value of the labeled region (clump or object).

--surfacebrightness
The surface brightness (in units of mag/arcsec$^2$) of the labeled region (objects or clumps). For more on the definition of the surface brightness, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

--serror
Error in measuring the surface brightness (the --surfacebrightness column). This column will use the value given to --spatialresolution in the processing (in pixels). For more on --spatialresolution, see Section 7.4.6.1 [MakeCatalog inputs and basic settings], page 409, and for the equation used to derive the surface brightness error, see Section 7.4.3.2 [Surface brightness error of each detection], page 400.

--areaxy
Similar to --area, when the clump or object is projected onto the first two dimensions. This is only available for 3-dimensional datasets. When working with Integral Field Unit (IFU) datasets, this projection onto the first two dimensions would be a narrow-band image.

--fwhm
The full width at half maximum (in units of pixels, along the semi-major axis) of the labeled region (object or clump). The maximum value is estimated from the mean of the top-three pixels with the highest values, see the description under --maximum. The number of pixels that have half the value of that maximum are then found (value in the --halfmaxarea column) and a radius is estimated from the area. See the description under --halfsumradius for more on converting area to radius along major axis.

Because of its non-parametric nature, this column is most reliable on clumps and should only be used in objects with great caution. This is because objects can have more than one clump (peak with true signal) and multiple peaks are not treated separately in objects, so the result of this column will be biased.

Also, because of its non-parametric nature, this FWHM it does not account for the PSF, and it will be strongly affected by noise if the object is faint. So when half the maximum value (which can be requested using the --maximum column) is too close to the local noise level (which can be requested using the --std column), the value returned in this column is meaningless (its just noise peaks which are randomly distributed over the area). You can therefore use the --maximum and --std columns to remove unreliable FWHMs. For example if a labeled region’s maximum is less than 2 times the sky standard deviation, the value will certainly be unreliable (half of that is 1σ!). For a more reliable value, this fraction should be around 4 (so half the maximum is 2σ).

--halfmaxarea
The number of pixels with values larger than half the maximum flux within the labeled region. This option is used to estimate --fwhm, so please read the notes there for the caveats and necessary precautions.
--halfmaxradius
The radius of region containing half the maximum flux within the labeled region.
This is just half the value reported by --fwhm.

--halfmaxsum
The sum of the pixel values containing half the maximum flux within the labeled region (or those that are counted in --halfmaxarea). This option uses the pixels within --fwhm, so please read the notes there for the caveats and necessary precautions.

--halfmaxsb
The surface brightness (in units of mag/arcsec$^2$) within the region that contains half the maximum value of the labeled region. For more on the definition of the surface brightness, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

--halfsumarea
The number of pixels that contain half the object or clump’s total sum of pixels (half the value in the --brightness column). To count this area, all the non-blank values associated with the given label (object or clump) will be sorted and summed in order (starting from the maximum), until the sum becomes larger than half the total sum of the label’s pixels.

This option is thus good for clumps (which are defined to have a single peak in their morphology), but for objects you should be careful: if the object includes multiple peaks/clumps at roughly the same level, then the area reported by this option will be distributed over all the peaks.

--halfsumsb
Surface brightness (in units of mag/arcsec$^2$) within the area that contains half the total sum of the label’s pixels (object or clump). For more on the definition of the surface brightness, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

This column just plugs in the values of half the value of the --brightness column and the --halfsumarea column, into the surface brightness equation. Therefore please see the description in --halfsumarea to understand the systematics of this column and potential biases.

--halfsumradius
Radius (in units of pixels) derived from the area that contains half the total sum of the label’s pixels (value reported by --halfsumarea). If the area is $A_h$ and the axis ratio is $q$, then the value returned in this column is $\sqrt{A_h/(\pi q)}$.

This option is a good measure of the concentration of the observed (after PSF convolution and noisy) object or clump, But as described below it underestimates the effective radius. Also, it should be used in caution with objects that may have multiple clumps. It is most reliable with clumps or objects that have one or zero clumps, see the note under --halfsumarea.

Recall that in general, for an ellipse with semi-major axis $a$, semi-minor axis $b$, and axis ratio $q = b/a$ the area $(A)$ is $A = \pi ab = \pi qa^2$. For a circle (where $q = 1$), this simplifies to the familiar $A = \pi a^2$. 
This option should not be confused with the *effective radius* for Sérsic profiles, commonly written as $r_e$. For more on the Sérsic profile and $r_e$, please see Section 8.1.1.4 [Galaxies], page 444. Therefore, when $r_e$ is meaningful for the target (the target is elliptically symmetric and can be parameterized as a Sérsic profile), $r_e$ should be derived from fitting the profile with a Sérsic function which has been convolved with the PSF. But from the equation above, you see that this radius is derived from the raw image’s labeled values (after convolution, with no parametric profile), so this column’s value will generally be (much) smaller than $r_e$, depending on the PSF, depth of the dataset, the morphology, or if a fraction of the profile falls on the edge of the image.

In other words, this option can only be interpreted as an effective radius if there is no noise and no PSF and the profile within the image extends to infinity (or a very large multiple of the effective radius) and it not near the edge of the image.

`--fracmaxarea1`  
`--fracmaxarea2`

Number of pixels brighter than the given fraction(s) of the maximum pixel value. For the maximum value, see the description of `--maximum` column. The fraction(s) are given through the `--fracmax` option (that can take two values) and is described in Section 7.4.6.1 [MakeCatalog inputs and basic settings], page 409. Recall that in `--halfmaxarea`, the fraction is fixed to 0.5. Hence, added with these two columns, you can sample three parts of the profile area.

`--fracmaxsum1`  
`--fracmaxsum2`

Sum of pixels brighter than the given fraction(s) of the maximum pixel value. For the maximum value, see the description of `--maximum` column below. The fraction(s) are given through the `--fracmax` option (that can take two values) and is described in Section 7.4.6.1 [MakeCatalog inputs and basic settings], page 409. Recall that in `--halfmaxsum`, the fraction is fixed to 0.5. Hence, added with these two columns, you can sample three parts of the profile’s sum of pixels.

`--fracmaxradius1`  
`--fracmaxradius2`

Radius (in units of pixels) derived from the area that contains the given fractions of the maximum valued pixel(s) of the label’s pixels (value reported by `--fracmaxarea1` or `--fracmaxarea2`). For the maximum value, see the description of `--maximum` column below. The fractions are given through the `--fracmax` option (that can take two values) and is described in Section 7.4.6.1 [MakeCatalog inputs and basic settings], page 409. Recall that in `--fwhm`, the fraction is fixed to 0.5. Hence, added with these two columns, you can sample three parts of the profile’s radius.

`--clumpsarea`  
[Objects] The total area of all the clumps in this object.

`--weightarea`

The area (number of pixels) used in the flux weighted position calculations.
--geoarea
The area of all the pixels labeled with an object or clump. Note that unlike
--area, pixel values are completely ignored in this column. For example, if a
pixel value is blank, it won’t be counted in --area, but will be counted here.

--geoareaxy
Similar to --geoarea, when the clump or object is projected onto the first two
dimensions. This is only available for 3-dimensional datasets. When working
with Integral Field Unit (IFU) datasets, this projection onto the first two
dimensions would be a narrow-band image.

-A
--semimajor
The pixel-value weighted root mean square (RMS) along the semi-major axis
of the profile (assuming it is an ellipse) in units of pixels. See Section 7.4.4
[Measuring elliptical parameters], page 404.

-B
--semiminor
The pixel-value weighted root mean square (RMS) along the semi-minor axis
of the profile (assuming it is an ellipse) in units of pixels. See Section 7.4.4
[Measuring elliptical parameters], page 404.

--axisratio
The pixel-value weighted axis ratio (semi-minor/semi-major) of the object or
clump.

-p
--positionangle
The pixel-value weighted angle of the semi-major axis with the first FITS axis
in degrees. See Section 7.4.4 [Measuring elliptical parameters], page 404.

--geosemimajor
The geometric (ignoring pixel values) root mean square (RMS) along the semi-
major axis of the profile, assuming it is an ellipse, in units of pixels.

--geosemiminor
The geometric (ignoring pixel values) root mean square (RMS) along the semi-
minor axis of the profile, assuming it is an ellipse, in units of pixels.

--geoaxisratio
The geometric (ignoring pixel values) axis ratio of the profile, assuming it is an
ellipse.

--geopositionangle
The geometric (ignoring pixel values) angle of the semi-major axis with the first
FITS axis in degrees.

Above, all of MakeCatalog’s single-valued measurements were listed. As mentioned in
the start of this section, MakeCatalog can also do multi-valued measurements per label.
Currently the only such measurement is the creation of spectra from 3D data cubes as
discussed below:
--spectrum
Generate a spectrum (measurement along the first two FITS dimensions) for each label when the input dataset is a 3D data cube. With this option, a separate table/spectrum will be generated for every label. If the output is a FITS file, each label’s spectrum will be written into an extension of that file with a standard name of SPECTRUM_NN (the label will be replaced with NN). If the output is a plain text file, each label’s spectrum will be written into a separate file with the suffix spec-NN.txt. See Section 7.4.6.4 [MakeCatalog output], page 427, for more on specifying MakeCatalog’s output file.

The spectra will contain one row for every slice (third FITS dimension) of the cube. Since the physical nature of the third dimension is different, two types of spectra (along with their errors) are measured: 1) Sum of values in each slice that only have the requested label. 2) Sum of values on the 2D projection of the whole label (the area of this projection can be requested with the --areaxy column above).

Labels can overlap when they are projected onto the first two FITS dimensions (the spatial domain). To help separate them, MakeCatalog does a third measurement on each slice: the area, sum of values and error of all pixels that belong to other labels but overlap with the 2D projection. This can be used to see how reliable the emission line measurement is (on the projected spectra) and also if multiple lines (labeled regions) belong to the same physical object.

--inbetweenints
Output will contain one row for all integers between 1 and the largest label in the input (irrespective of their existence in the input image). By default, MakeCatalog’s output will only contain rows with integers that actually corresponded to at least one pixel in the input dataset.

For example if the input’s only labeled pixel values are 11 and 13, MakeCatalog’s default output will only have two rows. If you use this option, it will have 13 rows and all the columns corresponding to integer identifiers that didn’t correspond to any pixel will be 0 or NaN (depending on context).

7.4.6.4 MakeCatalog output
After it has completed all the requested measurements (see Section 7.4.6.3 [MakeCatalog measurements], page 415), MakeCatalog will store its measurements in table(s). If an output filename is given (see --output in Section 4.1.2.1 [Input/Output options], page 157), the format of the table will be deduced from the name. When it isn’t given, the input name will be appended with a _cat suffix (see Section 4.9 [Automatic output], page 191) and its format will be determined from the --tableformat option, which is also discussed in Section 4.1.2.1 [Input/Output options], page 157. --tableformat is also necessary when the requested output name is a FITS table (recall that FITS can accept ASCII and binary tables, see Section 5.3 [Table], page 230).

By default (when --spectrum or --clumpscat aren’t called) only a single catalog/table will be created for the labeled “objects”.

- if --clumpscat is called, a secondary catalog/table will also be created for “clumps” (one of the outputs of the Segment program, for more on “objects” and “clumps”, see
Section 7.3 [Segment], page 381). In short, if you only have one labeled image, you don’t have to worry about clumps and just ignore this.

- When \texttt{--spectrum} is called, it is not mandatory to specify any single-valued measurement columns. In this case, the output will only be the spectra of each labeled region within a 3D datacube. For more, see the description of \texttt{--spectrum} in Section 7.4.6.3 [MakeCatalog measurements], page 415.

When possible, MakeCatalog will also measure the full input’s noise level (also known as surface brightness limit, see Section 7.4.3 [Quantifying measurement limits], page 398). Since these measurements are related to the noise and not any particular labeled object, they are stored as keywords in the output table. Furthermore, they are only possible when a standard deviation image has been loaded (done automatically for any column measurement that involves noise, for example \texttt{--sn} or \texttt{--std}). But if you just want the surface brightness limit and no noise-related column, you can use \texttt{--forcereadstd}. All these keywords start with \texttt{SBL} (for “surface brightness limit”) and are described below:

- \texttt{SBLSTD} Per-pixel standard deviation. If a \texttt{MEDSTD} keyword exists in the standard deviation dataset, then that value is directly used.
- \texttt{SBLNSIG} Sigma multiple for surface brightness limit (value you gave to \texttt{--sfmagnsigma}), used for \texttt{SBLMAGPX} and \texttt{SBLMAG}.
- \texttt{SBLMAGPX} Per-pixel surface brightness limit (in units of magnitudes/pixel).
- \texttt{SBLAREA} Area (in units of arcsec$^2$) used in \texttt{SBLMAG} (value you gave to \texttt{--sfmagarea}).
- \texttt{SBLMAG} Surface brightness limit of data calculated over \texttt{SBLAREA} (in units of mag/arcsec$^2$).

When any of the upper-limit measurements are requested, the input parameters for the upper-limit measurement are stored in the keywords starting with \texttt{UP}: \texttt{UPNSIGMA}, \texttt{UPNUMBER}, \texttt{UPRNGNAM}, \texttt{UPRNGSEE}, \texttt{UPSCMLTP}, \texttt{UPSCTOL}. These are primarily input arguments, so they correspond to the options with a similar name.

The full list of MakeCatalog’s options relating to the output file format and keywords are listed below. See Section 7.4.6.3 [MakeCatalog measurements], page 415, for specifying which columns you want in the final catalog.

\texttt{-C} \texttt{--clumpscat}

Do measurements on clumps and produce a second catalog (only devoted to clumps). When this option is given, MakeCatalog will also look for a secondary labeled dataset (identifying substructure) and produce a catalog from that. For more on the definition on “clumps”, see Section 7.3 [Segment], page 381.

When the output is a FITS file, the objects and clumps catalogs/tables will be stored as multiple extensions of one FITS file. You can use Section 5.3 [Table], page 230, to inspect the column meta-data and contents in this case. However, in plain text format (see Section 4.7.2 [Gnuastro text table format], page 185), it is only possible to keep one table per file. Therefore, if the output is a text file, two output files will be created, ending in \texttt{_o.txt} (for objects) and \texttt{_c.txt} (for clumps).
--noclumpsort
Don’t sort the clumps catalog based on object ID (only relevant with --clumpscat). This option will benefit the performance\textsuperscript{29} of MakeCatalog when it is run on multiple threads \textit{and} the position of the rows in the clumps catalog is irrelevant (for example you just want the number-counts).

MakeCatalog does all its measurements on each object independently and in parallel. As a result, while it is writing the measurements on each object’s clumps, it doesn’t know how many clumps there were in previous objects. Each thread will just fetch the first available row and write the information of clumps (in order) starting from that row. After all the measurements are done, by default (when this option isn’t called), MakeCatalog will reorder/permute the clumps catalog to have both the object and clump ID in an ascending order.

If you would like to order the catalog later (when its a plain text file), you can run the following command to sort the rows by object ID (and clump ID within each object), assuming they are respectively the first and second columns:

\texttt{
$ awk '!/^#/\}' out.c.txt | sort -g -k1,1 -k2,2
}

--sfmagnsigma=FLT
Value to multiply with the median standard deviation (from a MEDSTD keyword in the Sky standard deviation image) for estimating the surface brightness limit. Note that the surface brightness limit is only reported when a standard deviation image is read, in other words a column using it is requested (for example --sn) or --forcereadstd is called.

This value is a per-pixel value, not per object/clump and is not found over an area or aperture, like the common 5\(\sigma\) values that are commonly reported as a measure of depth or the upper-limit measurements (see Section 7.4.3 [Quantifying measurement limits], page 398).

--sfmagarea=FLT
Area (in arc-seconds squared) to convert the per-pixel estimation of --sfmagnsigma in the comments section of the output tables. Note that the surface brightness limit is only reported when a standard deviation image is read, in other words a column using it is requested (for example --sn) or --forcereadstd is called.

Note that this is just a unit conversion using the World Coordinate System (WCS) information in the input’s header. It does not actually do any measurements on this area. For random measurements on any area, please use the upper-limit columns of MakeCatalog (see the discussion on upper-limit measurements in Section 7.4.3 [Quantifying measurement limits], page 398).

7.5 Match
Data can come from different telescopes, filters, software and even different configurations for a single software. As a result, one of the primary things to do after generating

\textsuperscript{29} The performance boost due to --noclumpsort can only be felt when there are a huge number of objects. Therefore, by default the output is sorted to avoid miss-understandings or bugs in the user’s scripts when the user forgets to sort the outputs.
catalogs from each of these sources (for example with Section 7.4 \textit{[MakeCatalog]}, page 392), is to find which sources in one catalog correspond to which in the other(s). In other words, to ‘match’ the two catalogs with each other.

Gnuastro’s Match program is in charge of such operations. The nearest objects in the two catalogs, within the given aperture, will be found and given as output. The aperture can be a circle or an ellipse with any orientation.

\section*{Matching algorithms}

Matching involves two catalogs, let’s call them catalog A (with N rows) and catalog B (with M rows). The most basic matching algorithm that immediately comes to mind is this: for each row in A (let’s call it \(A_i\)), go over all the rows in B (\(B_j\), where \(0 < j < M\)) and calculate the distance \(|B_j - A_i|\). If this distance is less than a certain acceptable distance threshold (or radius, or aperture), consider \(A_i\) and \(B_j\) as a match.

This basic parsing algorithm is very computationally expensive: \(N \times M\) distances have to measured, and calculating the distance requires a square root and power of 2: in 2 dimensions it would be \(\sqrt{(B_{ix} - A_{ix})^2 + (B_{iy} - A_{iy})^2}\). If an elliptical aperture is necessary, it can even get more complicated, see Section 8.1.1.1 \textit{[Defining an ellipse and ellipsoid]}, page 440. Such operations are not simple, and will consume many cycles of your CPU! As a result, this basic algorithm will become terribly slow as your datasets grow in size. For example when N or M exceed hundreds of thousands (which is common in the current days with datasets like the European Space Agency’s Gaia mission). Therefore that basic parsing algorithm will take too much time and more efficient ways to \textit{find the nearest neighbor} need to be found. Gnuastro’s Match currently has algorithms for finding the nearest neighbor:

\subsection*{Sort-based}

In this algorithm, we will use a moving window over the sorted datasets:

1. Sort the two datasets by their first coordinate. Therefore \(A_i < A_j\) (when \(i < j\); only in first coordinate), and similarly, sort the elements of B based on the first coordinate.
2. Use the radial distance threshold to define the width of a moving interval over both A and B. Therefore, with a single parsing of both simultaneously, for each A-point, we can find all the elements in B that are sufficiently near to it (within the requested aperture).

This method has some caveats: 1) It requires sorting, which can again be slow on large numbers. 2) It can only be done on a single CPU thread! So it can’t benefit from the modern CPUs with many threads. 3) There is no way to preserve intermediate information for future matches, for example this can greatly help when one of the matched datasets is always the same. To use this sorting method in Match, use \texttt{--kdtree=disable}.

\subsection*{k-d tree based}

The k-d tree concept is much more abstract, but powerful (addressing all the caveats of the sort-based method described above.). In short a k-d tree is a partitioning of a k-dimensional space (“k” is just a place-holder, so together with “d” for dimension, “k-d” means “any number of dimensions”!). The k-d tree of table A is another table with the same number of rows, but only two integer columns: the integers contain the row indexes (counting from zero) of the
left and right “branch” (in the “tree”) of that row. With a k-d tree we can find
the nearest point with much fewer (statistically) checks, compared to always
parsing from the top-down. For more on the k-d tree concept and Gnuastro’s
implementation, please see Section 11.3.19 [K-d tree (kdtree.h)], page 625.
When given two catalogs (like the command below), Gnuastro’s Match will
internally construct a k-d tree for catalog A (the first catalog given to it) and
use the k-d tree of A, for finding the nearest B-point(s) to each A-point (this
is done in parallel on all available CPU threads, unless you specify a certain
number of threads to use with --numthreads, see Section 4.4 [Multi-threaded
operations], page 175)

$ astmatch A.fits --ccol1=ra,dec B.fits --ccol2=RA,DEC \
   --aperture=1/3600

However, optionally, you can also build the k-d tree of A and save it into a file,
with a separate call to Match, like below

$ astmatch A.fits --ccol1=ra,dec --kdtree=build \
   --output=A-kdtree.fits

This external k-d tree (A-kdtree.fits) can be fed to Match later (to avoid
having to reconstruct it every time you want to match a new catalog with A)
like below for matching both B.fits and C.fits with A.fits. Note that the
same --kdtree option above, is now given the file name of the k-d tree, instead
of build.

$ astmatch A.fits --ccol1=ra,dec --kdtree=A-kdtree.fits \ 
   B.fits --ccol2=RA,DEC --aperture=1/3600 \ 
   --output=A-B.fits

$ astmatch A.fits --ccol1=ra,dec --kdtree=A-kdtree.fits \ 
   C.fits --ccol2=RA,DEC --aperture=1/3600 \ 
   --output=A-C.fits

Irrespective of how the k-d tree is made ready (by importing or by constructing
internally), it will be used to find the nearest A-point to each B-point. The k-d
tree is parsed independently (on different CPU threads) for each row of B.
There is just one technical issue however: when there is no neighbor within the
acceptable distance of the k-d tree, it is forced to parse all elements to confirm
that there is no match! Therefore if one catalog only covers a small portion (in
the coordinate space) of the other catalog, the k-d tree algorithm will be forced
to parse the full k-d tree for the majority of points! This will dramatically
decrease the running speed of Match. Therefore, Match first divides the range
of the first input in all its dimensions into bins that have a width of the requested
aperture (similar to a histogram), and will only do the k-d tree based search
when the point in catalog B actually falls within a bin that has at least one
element in A.

Above, we described different ways of finding the \(A_i\) that is nearest to each \(B_j\). But
this isn’t the whole matching process! Let’s go ahead with a “basic” description of what
happens next... You may be tempted to remove \(A_i\) from the search of matches for \(B_k\) (where
\(k > j\)). Therefore, as you go down B (and more matches are found), you have to calculate
less distances (there are fewer elements in A that remain to be checked). However, this will
introduce an important bias: \( A_i \) may actually be closer to \( B_k \) than to \( B_j \)! But because \( B_j \) happened to be before \( B_k \) in your table, \( A_i \) was removed from the potential search domain of \( B_k \). The good match \( (B_k \text{ with } A_i) \) will therefore be lost, and replaced by a false match between \( B_j \) and \( A_i \)!

In a single-dimensional match, this bias depends on the sorting of your two datasets (leading to different matches if you shuffle your datasets). But it will get more complex as you add dimensionality. For example catalogs derived from 2D images or 3D cubes, where you have 2 and 3 different coordinates for each point.

To address this problem, in Gnuastro (the Match program, or the matching functions of the library) similar to above, we first parse over the elements of B. But we won’t associate the first nearest-neighbor with a match! Instead, we will use an array (with the number of rows in A, let’s call it “B-in-A”) to keep the list of all nearest element(s) in B that match each A-point. Once all the points in B are parsed, each A-point in B-in-A will (possibly) have a sorted list of B-points (there may be multiple B-points that fall within the acceptable aperture of each A-point). In the previous example, the \( i \) element (corresponding to \( A_i \)) of B-in-A will contain the following list of B-points: \( B_j \) and \( B_k \).

A new array (with the number of points in B, let’s call it A-in-B) is then used to find the final match. We parse over B-in-A (that was completed above), and from it extract the nearest B-point to each A-point (\( B_k \text{ for } A_i \) in the example above). If this is the first A-point that is found for this B-point, then we put this A-point into A-in-B (in the example above, element \( k \) is filled with \( A_k \)). If another A-point was previously found for this B-point, then the distance of the two A-points to that B-point are compared, and the A-point with the smaller distance is kept in A-in-B. This will give us the best match between the two catalogs, independent of any sorting issues. Both the B-in-A and A-in-B will also keep the distances, so distances are only measured once.

In summary, here are the points to consider when selecting an algorithm, or the order of your inputs (for optimal speed, the match will be the same):

- For larger datasets, the k-d tree based method (when running on all threads possible) is much more faster than the classical sort-based method.
- The k-d tree is constructed for the first input table and the multi-threading is done on the rows of the second input table. The construction of a larger dataset’s k-d tree will take longer, but multi-threading will work better when you have more rows. As a result, the optimal way to place your inputs is to give the smaller input table (with fewer rows) as the first argument (so its k-d tree is constructed), and the larger table as the second argument (so its rows are checked in parallel).
- If you always need to match against one catalog (that is large!), the k-d tree construction itself can take a significant fraction of the running time. Therefore you can save its k-d tree into a file and simply give it to later calls, like the example given in the description of the k-d algorithm mentioned above.

### 7.5.2 Invoking Match

When given two catalogs, Match finds the rows that are nearest to each other within an input aperture. The executable name is `astmatch` with the following general template

\[
\$ astmatch \text{ [OPTION ...]} \text{ input-1 input-2}
\]
One line examples:

```bash
## 1D wavelength match (within 5 angstroms) of the two inputs.
## The wavelengths are in the 5th and 10th columns respectively.
$ astmatch --aperture=5e-10 --ccol1=5 --ccol2=10 in1.fits in2.txt

## Find the row that is closest to (RA,DEC) of (12.3456,6.7890)
## with a maximum distance of 1 arcseconds (1/3600 degrees).
## The coordinates can also be given in sexagesimal.
$ astmatch input1.txt --ccol1=ra,dec --coord=12.3456,6.7890 \ --aperture=1/3600

## Find matching rows of two catalogs with a circular aperture
## of width 2 (same unit as position columns: pixels in this case).
$ astmatch input1.txt input2.fits --aperture=2 \ --ccol1=X,Y --ccol2=IMG_X,IMG_Y

## Similar to before, but the output is created by merging various
## columns from the two inputs: columns 1, RA, DEC from the first
## input, followed by all columns starting with 'MAG' and the 'BRG'
## column from second input and the 10th column from first input.
$ astmatch input1.txt input2.fits --aperture=1/3600 \ --ccol1=ra,dec --ccol2=RAJ2000,DEJ2000 \ --outcols=a1,aRA,aDEC,b/^MAG/,bBRG,a10

## Assuming both inputs have the same column metadata (same name
## and numeric type), the output will contain all the rows of the
## first input, appended with the non-matching rows of the second
## input (good when you need to merge multiple catalogs that
## may have matching items, which you don't want to repeat).
$ astmatch input1.fits input2.fits --aperture=1/3600 --notmatched --outcols=_all

## Match the two catalogs within an elliptical aperture of 1 and 2
## arc-seconds along RA and Dec respectively.
$ astmatch --aperture=1/3600,2/3600 in1.fits in2.txt

## Match the RA and DEC columns of the first input with the RA_D
## and DEC_D columns of the second within a 0.5 arcseconds aperture.
$ astmatch --ccol1=RA,DEC --ccol2=RA_D,DEC_D --aperture=0.5/3600 \ in1.fits in2.fits

## Match in 3D (RA, Dec and Wavelength).
$ astmatch --ccol1=2,3,4 --ccol2=2,3,4 -a0.5/3600,0.5/3600,5e-10 \ in1.fits in2.txt
```

Match will find the rows that are nearest to each other in two catalogs (given some
cordinate columns). Alternatively, it can construct the k-d tree of one catalog to save
in a FITS file for future matching of the same catalog with many others. To understand the inner working of Match and its algorithms, see Section 7.5.1 [Matching algorithms], page 430.

When matching, two catalogs are necessary for input. But for constructing a k-d tree, only a single catalog should be given. The input tables can be plain text tables or FITS tables, for more see Section 4.7 [Tables], page 183. But other ways of feeding inputs area also supported:

- The first catalog can also come from the standard input (for example a pipe that feeds the output of a previous command to Match, see Section 4.1.4 [Standard input], page 167);
- When you only want to match one point with another catalog, you can use the --coord option to avoid creating a file for the second input catalog.

Match follows the same basic behavior of all Gnuastro programs as fully described in Chapter 4 [Common program behavior], page 152. If the first input is a FITS file, the common --hdu option (see Section 4.1.2.1 [Input/Output options], page 157) should be used to identify the extension. When the second input is FITS, the extension must be specified with --hdu2.

When --quiet is not called, Match will print its various processing phases (including the number of matches found) in standard output (on the command-line). When matches are found, by default, two tables will be output (if in FITS format, as two HDUs). Each output table will contain the re-arranged rows of the respective input table. In other words, both tables will have the same number of rows, and row N in both corresponds to the 10th match between the two. If no matches are found, the columns of the output table(s) will have zero rows (with proper meta-data). The output format can be changed with the following options:

- --outcols: The output will be a single table with rows chosen from either of the two inputs in any order.
- --notmatched: The output tables will contain the rows that didn’t match between the two tables. If called with --outcols, the output will be a single table with all non-matched rows of both tables.
- --logasoutput: The output will be a single table with the contents of the log file, see below.

If no output file name is given with the --output option, then automatic output Section 4.9 [Automatic output], page 191, will be used to determine the output name(s). Depending on --tableformat (see Section 4.1.2.1 [Input/Output options], page 157), the output will be a (possibly multi-extension) FITS file or (possibly two) plain text file(s). Generally, giving a filename to --output is recommended.

When the --log option is called (see Section 4.1.2.3 [Operating mode options], page 162), and there was a match, Match will also create a file named astmatch.fits (or astmatch.txt, depending on --tableformat, see Section 4.1.2.1 [Input/Output options], page 157) in the directory it is run in. This log table will have three columns. The first and second columns show the matching row/record number (counting from 1) of the first and second input catalogs respectively. The third column is the distance between the two matched positions. The units of the distance are the same as the given coordinates (given
the possible ellipticity, see description of \texttt{--aperture} below). When \texttt{--logasoutput} is called, no log file (with a fixed name) will be created. In this case, the output file (possibly given by the \texttt{--output} option) will have the contents of this log file.

\textbf{--log isn’t thread-safe:} As described above, when \texttt{--logasoutput} is not called, the Log file has a fixed name for all calls to Match. Therefore if a separate log is requested in two simultaneous calls to Match in the same directory, Match will try to write to the same file. This will cause problems like unreasonable log file, undefined behavior, or a crash. Remember that \texttt{--log} is mainly intended for debugging purposes, if you want the log file with a specific name, simply use \texttt{--logasoutput} (which will also be faster, since no arranging of the input columns is necessary).

\texttt{-H STR}  
\texttt{--hdu2=STR}  
The extension/HDU of the second input if it is a FITS file. When it isn’t a FITS file, this option’s value is ignored. For the first input, the common option \texttt{--hdu} must be used.

\texttt{-k STR}  
\texttt{--kdtree=STR}  
Select the algorithm and/or the way to construct or import the k-d tree. A summary of the four acceptable strings for this option are described here for completeness. However, for a much more detailed discussion on Match’s algorithms with examples, see Section 7.5.1 [Matching algorithms], page 430.

\textbf{internal} Construct a k-d tree for the first input internally (within the same run of Match), and parallelize over the rows of the second to find the nearest points. This is the default algorithm/method used by Match (when this option isn’t called).

\textbf{build} Only construct a k-d tree of a single input and abort. The name of the k-d tree is value to \texttt{--output}.

\textbf{CUSTOM-FITS-FILE}  
Use the given FITS file as a k-d tree (that was previously constructed with Match itself) of the first input, and don’t construct any k-d tree internally. The FITS file should have two columns with an unsigned 32-bit integer data type and a \texttt{KDTROOT} keyword that contains the index of the root of the k-d tree. For more on Gnuastro’s k-d tree format, see Section 11.3.19 [K-d tree (kdtree.h)], page 625.

\textbf{disable} Don’t use the k-d tree algorithm for finding the nearest neighbor, instead, use the sort-based method.

\texttt{--kdtreehdu=STR}  
The HDU of the FITS file, when a FITS file is given to the \texttt{--kdtree} option that was described above.
--outcols=STR[,STR,[...]]

Columns (from both inputs) to write into a single matched table output. The value to --outcols must be a comma-separated list of column identifiers (number or name, see Section 4.7.3 [Selecting table columns], page 188). The expected format depends on --notmatched and explained below. By default (when --nomatched is not called), the number of rows in the output will be equal to the number of matches. However, when --notmatched is called, all the rows (from the requested columns) of the first input are placed in the output, and the not-matched rows of the second input are inserted afterwards (useful when you want to merge unique entries of multiple catalogs into one).

Default (only matching rows)

The first character of each string specifies the input catalog: a for the first and b for the second. The rest of the characters of the string will be directly used to identify the proper column(s) in the respective table. See Section 4.7.3 [Selecting table columns], page 188, for how columns can be specified in Gnuastro.

For example the output of --outcols=a1,bRA,bDEC will have three columns: the first column of the first input, along with the RA and DEC columns of the second input.

If the string after a or b is _all, then all the columns of the respective input file will be written in the output. For example the command below will print all the input columns from the first catalog along with the 5th column from the second:

```
$ astmatch a.fits b.fits --outcols=a_all,b5
```

_all can be used multiple times, possibly on both inputs. Tip: if an input’s column is called _all (an unlikely name!) and you don’t want all the columns from that table the output, use its column number to avoid confusion.

Another example is given in the one-line examples above. Compared to the default case (where two tables with all their columns) are saved separately, using this option is much faster: it will only read and re-arrange the necessary columns and it will write a single output table. Combined with regular expressions in large tables, this can be a very powerful and convenient way to merge various tables into one.

When --coord is given, no second catalog will be read. The second catalog will be created internally based on the values given to --coord. So column names aren’t defined and you can only request integer column numbers that are less than the number of coordinates given to --coord. For example if you want to find the row matching RA of 1.2345 and Dec of 6.7890, then you should use --coord=1.2345,6.7890. But when using --outcols, you can’t give bRA, or b25.
With **--notmatched**

Only the column names/numbers should be given (for example **--outcols=RA,DEC,MAGNITUDE**). It is assumed that both input tables have the requested column(s) and that the numerical data types of each column in each input (with same name) is the same as the corresponding column in the other. Therefore if one input has a MAGNITUDE column with a 32-bit floating point type, but the MAGNITUDE column of the other is 64-bit floating point, Match will crash with an error. The metadata of the columns will come from the first input.

As an example, let's assume `input1.txt` and `input2.fits` each have a different number of columns and rows. However, they both have the RA (64-bit floating point), DEC (64-bit floating point) and MAGNITUDE (32-bit floating point) columns. If `input1.txt` has 100 rows and `input2.fits` has 300 rows (such that 50 of them match within 1 arcsec of the first), then the output of the command above will have \(100 + (300 - 50) = 350\) rows and only three columns. Other columns in each catalog, which may be different, are ignored.

```
$ astmatch input1.txt --ccol1=RA,DEC \
  input2.fits --ccol2=RA,DEC \
  --aperture=1/3600 \
  --notmatched --outcols=RA,DEC,MAGNITUDE
```

**--logasoutput**

The output file will have the contents of the log file: indexes in the two catalogs that match with each other along with their distance, see description of the log file above.

When this option is called, a separate log file will not be created and the output will not contain any of the input columns (either as two tables containing the re-arranged columns of each input, or a single table mixing columns), only their indices in the log format.

**--notmatched**

Write the non-matching rows into the outputs, not the matched ones. By default, this will produce two output tables, that will not necessarily have the same number of rows. However, when called with **--outcols**, its possible to import non-matching rows of the second into the first. See the description of **--outcols** for more.

**-c INT/STR[,INT/STR]
--ccol1=INT/STR[,INT/STR]**

The coordinate columns of the first input. The number of dimensions for the match is determined by the number of comma-separated values given to this option. The values can be the column number (counting from 1), exact column name or a regular expression. For more, see Section 4.7.3 [Selecting table columns], page 188. See the one-line examples above for some usages of this option.
-C INT/STR[,INT/STR]
--ccol2=INT/STR[,INT/STR]

The coordinate columns of the second input. See the example in --ccol1 for more.

-d FLT[,FLT]
--coord=FLT[,FLT]

Manually specify the coordinates to match against the given catalog. With this option, Match will not look for a second input file/table and will directly use the coordinates given to this option. When the coordinates are RA and Dec, the comma-separated values can either be in degrees (a single number), or sexagesimal (_h_m_ for RA, _d_m_ for Dec, or _:_:_ for both).

When this option is called, the output changes in the following ways: 1) when --outcols is specified, for the second input, it can only accept integer numbers that are less than the number of values given to this option, see description of that option for more. 2) By default (when --outcols isn’t used), only the matching row of the first table will be output (a single file), not two separate files (one for each table).

This option is good when you have a (large) catalog and only want to match a single coordinate to it (for example to find the nearest catalog entry to your desired point). With this option, you can write the coordinates on the command-line and thus avoid the need to make a single-row file.

-a FLT[,FLT[,FLT]]
--aperture=FLT[,FLT[,FLT]]

Parameters of the aperture for matching. The values given to this option can be fractions, for example when the position columns are in units of degrees, 1/3600 can be used to ask for one arc-second. The interpretation of the values depends on the requested dimensions (determined from --ccol1 and --ccol2) and how many values are given to this option.

When multiple objects are found within the aperture, the match is defined as the nearest one. In a multi-dimensional dataset, when the aperture is a general ellipse or ellipsoid (and not a circle or sphere), the distance is calculated in the elliptical space along the major axis. For the definition of this distance, see $r_{cl}$ in Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440.

1D match The aperture/interval can only take one value: half of the interval around each point (maximum distance from each point).

2D match In a 2D match, the aperture can be a circle, an ellipse aligned in the axes or an ellipse with a rotated major axis. To simply the usage, you can determine the shape based on the number of free parameters for each.

1 number For example --aperture=2. The aperture will be a circle of the given radius. The value will be in the same units as the columns in --ccol1 and --ccol2).

2 numbers For example --aperture=3,4e-10. The aperture will be an ellipse (if the two numbers are different) with the
Chapter 7: Data analysis

respective value along each dimension. The numbers are in units of the first and second axis. In the example above, the semi-axis value along the first axis will be 3 (in units of the first coordinate) and along the second axis will be $4 \times 10^{-10}$ (in units of the second coordinate). Such values can happen if you are comparing catalogs of a spectra for example. If more than one object exists in the aperture, the nearest will be found along the major axis as described in Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440.

3 numbers For example --aperture=2,0.6,30. The aperture will be an ellipse (if the second value is not 1). The first number is the semi-major axis, the second is the axis ratio and the third is the position angle (in degrees). If multiple matches are found within the ellipse, the distance (to find the nearest) is calculated along the major axis in the elliptical space, see Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440.

3D match The aperture (matching volume) can be a sphere, an ellipsoid aligned on the three axises or a general ellipsoid rotated in any direction. To simplify the usage, the shape can be determined based on the number of values given to this option.

1 number For example --aperture=3. The matching volume will be a sphere of the given radius. The value is in the same units as the input coordinates.

3 numbers For example --aperture=4,5,6e-10. The aperture will be a general ellipsoid with the respective extent along each dimension. The numbers must be in the same units as each axis. This is very similar to the two number case of 2D inputs. See there for more.

6 numbers For example --aperture=4,0.5,0.6,10,20,30. The numbers represent the full general ellipsoid definition (in any orientation). For the definition of a general ellipsoid, see Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440. The first number is the semi-major axis. The second and third are the two axis ratios. The last three are the three Euler angles in units of degrees in the ZXZ order as fully described in Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440.
8 Modeling and fitting

In order to fully understand observations after initial analysis on the image, it is very important to compare them with the existing models to be able to further understand both the models and the data. The tools in this chapter create model galaxies and will provide 2D fittings to be able to understand the detections.

8.1 MakeProfiles

MakeProfiles will create mock astronomical profiles from a catalog, either individually or together in one output image. In data analysis, making a mock image can act like a calibration tool, through which you can test how successfully your detection technique is able to detect a known set of objects. There are commonly two aspects to detecting: the detection of the fainter parts of bright objects (which in the case of galaxies fade into the noise very slowly) or the complete detection of an over-all faint object. Making mock galaxies is the most accurate (and idealistic) way these two aspects of a detection algorithm can be tested. You also need mock profiles in fitting known functional profiles with observations.

MakeProfiles was initially built for extra galactic studies, so currently the only astronomical objects it can produce are stars and galaxies. We welcome the simulation of any other astronomical object. The general outline of the steps that MakeProfiles takes are the following:

1. Build the full profile out to its truncation radius in a possibly over-sampled array.
2. Multiply all the elements by a fixed constant so its total magnitude equals the desired total magnitude.
3. If --individual is called, save the array for each profile to a FITS file.
4. If --nomerged is not called, add the overlapping pixels of all the created profiles to the output image and abort.

Using input values, MakeProfiles adds the World Coordinate System (WCS) headers of the FITS standard to all its outputs (except PSF images!). For a simple test on a set of mock galaxies in one image, there is no need for the third step or the WCS information.

However in complicated simulations like weak lensing simulations, where each galaxy undergoes various types of individual transformations based on their position, those transformations can be applied to the different individual images with other programs. After all the transformations are applied, using the WCS information in each individual profile image, they can be merged into one output image for convolution and adding noise.

8.1.1 Modeling basics

In the subsections below, first a review of some very basic information and concepts behind modeling a real astronomical image is given. You can skip this subsection if you are already sufficiently familiar with these concepts.

8.1.1.1 Defining an ellipse and ellipsoid

The PSF, see Section 8.1.1.2 [Point spread function], page 442, and galaxy radial profiles are generally defined on an ellipse. Therefore, in this section we’ll start defining an ellipse on a pixelated 2D surface. Labeling the major axis of an ellipse $a$, and its minor axis with
b, the axis ratio is defined as: \( q \equiv b/a \). The major axis of an ellipse can be aligned in any direction, therefore the angle of the major axis with respect to the horizontal axis of the image is defined to be the position angle of the ellipse and in this book, we show it with \( \theta \).

Our aim is to put a radial profile of any functional form \( f(r) \) over an ellipse. Hence we need to associate a radius/distance to every point in space. Let’s define the radial distance \( r_{el} \) as the distance on the major axis to the center of an ellipse which is located at \( i_c \) and \( j_c \) (in other words \( r_{el} \equiv a \)). We want to find \( r_{el} \) of a point located at \((i,j)\) (in the image coordinate system) from the center of the ellipse with axis ratio \( q \) and position angle \( \theta \). First the coordinate system is rotated\(^1\) by \( \theta \) to get the new rotated coordinates of that point \((i_r,j_r)\):

\[
\begin{align*}
i_r(i,j) &= +(i_c - i) \cos \theta + (j_c - j) \sin \theta \\
j_r(i,j) &= -(i_c - i) \sin \theta + (j_c - j) \cos \theta
\end{align*}
\]

Recall that an ellipse is defined by \((i_r/a)^2 + (j_r/b)^2 = 1\) and that we defined \( r_{el} \equiv a \). Hence, multiplying all elements of the ellipse definition with \( r_{el}^2 \) we get the elliptical distance at this point point located: \( r_{el} = \sqrt{i_r^2 + (j_r/q)^2} \). To place the radial profiles explained below over an ellipse, \( f(r_{el}) \) is calculated based on the functional radial profile desired.

An ellipse in 3D, or an ellipsoid (https://en.wikipedia.org/wiki/Ellipsoid), can be defined following similar principles as before. Labeling the major (largest) axis length as \( a \), the second and third (in a right-handed coordinate system) axis lengths can be labeled as \( b \) and \( c \). Hence we have two axis ratios: \( q_1 \equiv b/a \) and \( q_2 \equiv c/a \). The orientation of the ellipsoid can be defined from the orientation of its major axis. There are many ways to define 3D orientation and order matters. So to be clear, here we use the ZXZ (or \( Z_1X_2Z_3 \)) proper Euler angles (https://en.wikipedia.org/wiki/Euler_angles) to define the 3D orientation. In short, when a point is rotated in this order, we first rotate it around the \( Z \) axis (third axis) by \( \alpha \), then about the (rotated) \( X \) axis by \( \beta \) and finally about the (rotated) \( Z \) axis by \( \gamma \).

Following the discussion in Section 6.4.2 [Merging multiple warpings], page 331, we can define the full rotation with the following matrix multiplication. However, here we are rotating the coordinates, not the point. Therefore, both the rotation angles and rotation order are reversed. We are also not using homogeneous coordinates (see Section 6.4.1 [Warping basics], page 329) since we aren’t concerned with translation in this context:

\[
\begin{bmatrix}
i_r \\
j_r \\
k_r
\end{bmatrix} =
\begin{bmatrix}
cos \gamma & sin \gamma & 0 \\
-sin \gamma & cos \gamma & 0 \\
0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
1 & 0 & 0 \\
0 & cos \beta & sin \beta \\
0 & -sin \beta & cos \beta
\end{bmatrix}
\begin{bmatrix}
cosa & sina & 0 \\
-sina & cosa & 0 \\
0 & 0 & 1
\end{bmatrix}
\begin{bmatrix}
i_c - i \\
j_c - j \\
k_c - k
\end{bmatrix}
\]

Recall that an ellipsoid can be characterized with \((i_r/a)^2 + (j_r/b)^2 + (k_r/c)^2 = 1\), so similar to before \((r_{el} \equiv a)\), we can find the ellipsoidal radius at pixel \((i,j,k)\) as: \( r_{el} = \sqrt{i_r^2 + (j_r/q_1)^2 + (k_r/q_2)^2} \).

\(^1\) Do not confuse the signs of \( sin \) with the rotation matrix defined in Section 6.4.1 [Warping basics], page 329. In that equation, the point is rotated, here the coordinates are rotated and the point is fixed.
MakeProfiles builds the profile starting from the nearest element (pixel in an image) in the dataset to the profile center. The profile value is calculated for that central pixel using monte carlo integration, see Section 8.1.1.5 [Sampling from a function], page 444. The next pixel is the next nearest neighbor to the central pixel as defined by \( r_{el} \). This process goes on until the profile is fully built up to the truncation radius. This is done fairly efficiently using a breadth first parsing strategy\(^2\) which is implemented through an ordered linked list.

Using this approach, we build the profile by expanding the circumference. Not one more extra pixel has to be checked (the calculation of \( r_{el} \) from above is not cheap in CPU terms). Another consequence of this strategy is that extending MakeProfiles to three dimensions becomes very simple: only the neighbors of each pixel have to be changed. Everything else after that (when the pixel index and its radial profile have entered the linked list) is the same, no matter the number of dimensions we are dealing with.

### 8.1.1.2 Point spread function

Assume we have a ‘point’ source, or a source that is far smaller than the maximum resolution (a pixel). When we take an image of it, it will ‘spread’ over an area. To quantify that spread, we can define a ‘function’. This is how the “point spread function” or the PSF of an image is defined.

This ‘spread’ can have various causes, for example in ground-based astronomy, due to the atmosphere. In practice we can never surpass the ‘spread’ due to the diffraction of the telescope aperture (even in Space!). Various other effects can also be quantified through a PSF. For example, the simple fact that we are sampling in a discrete space, namely the pixels, also produces a very small ‘spread’ in the image.

Convolution is the mathematical process by which we can apply a ‘spread’ to an image, or in other words blur the image, see Section 6.3.1.1 [Convolution process], page 307. The Brightness of an object should remain unchanged after convolution, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394. Therefore, it is important that the sum of all the pixels of the PSF be unity. The PSF image also has to have an odd number of pixels on its sides so one pixel can be defined as the center.

In MakeProfiles, the PSF can be set by the two methods explained below:

**Parametric functions**

A known mathematical function is used to make the PSF. In this case, only the parameters to define the functions are necessary and MakeProfiles will make a PSF based on the given parameters for each function. In both cases, the center of the profile has to be exactly in the middle of the central pixel of the PSF (which is automatically done by MakeProfiles). When talking about the PSF, usually, the full width at half maximum or FWHM is used as a scale of the width of the PSF.

**Gaussian**

In the older papers, and to a lesser extent even today, some researchers use the 2D Gaussian function to approximate the PSF of ground based images. In its most general form, a Gaussian function can be written as:

\[^2\) http://en.wikipedia.org/wiki/Breadth-first_search\]
\[ f(r) = a \exp \left( \frac{-(x - \mu)^2}{2\sigma^2} \right) + d \]

Since the center of the profile is pre-defined, \( \mu \) and \( d \) are constrained. \( a \) can also be found because the function has to be normalized. So the only important parameter for MakeProfiles is the \( \sigma \). In the Gaussian function we have this relation between the FWHM and \( \sigma \):

\[ \text{FWHM}_g = 2\sqrt{2\ln 2} \sigma \approx 2.35482\sigma \]

**Moffat**

The Gaussian profile is much sharper than the images taken from stars on photographic plates or CCDs. Therefore in 1969, Moffat proposed this functional form for the image of stars:

\[ f(r) = a \left[ 1 + \left( \frac{r}{\alpha} \right)^2 \right]^{-\beta} \]

Again, \( a \) is constrained by the normalization, therefore two parameters define the shape of the Moffat function: \( \alpha \) and \( \beta \). The radial parameter is \( \alpha \) which is related to the FWHM by

\[ \text{FWHM}_m = 2\alpha \sqrt{2^{1/\beta} - 1} \]

Comparing with the PSF predicted from atmospheric turbulence theory with a Moffat function, Trujillo et al.\(^3\) claim that \( \beta \) should be 4.765. They also show how the Moffat PSF contains the Gaussian PSF as a limiting case when \( \beta \to \infty \).

An input FITS image

An input image file can also be specified to be used as a PSF. If the sum of its pixels are not equal to 1, the pixels will be multiplied by a fraction so the sum does become 1.

Gnuastro has tools to extract the non-parametric (extended) PSF of any image as a FITS file (assuming there are a sufficient number of stars in it), see Section 2.4 [Building the extended PSF], page 97. This method is not perfect (will have noise if you don’t have many stars), but it is the actual PSF of the data that is not forced into any parametric form.

While the Gaussian is only dependent on the FWHM, the Moffat function is also dependent on \( \beta \). Comparing these two functions with a fixed FWHM gives the following results:

- Within the FWHM, the functions don’t have significant differences.

• For a fixed FWHM, as \( \beta \) increases, the Moffat function becomes sharper.
• The Gaussian function is much sharper than the Moffat functions, even when \( \beta \) is large.

### 8.1.1.3 Stars

In MakeProfiles, stars are generally considered to be a point source. This is usually the case for extra galactic studies, where nearby stars are also in the field. Since a star is only a point source, we assume that it only fills one pixel prior to convolution. In fact, exactly for this reason, in astronomical images the light profiles of stars are one of the best methods to understand the shape of the PSF and a very large fraction of scientific research is performed by assuming the shapes of stars to be the PSF of the image.

### 8.1.1.4 Galaxies

Today, most practitioners agree that the flux of galaxies can be modeled with one or a few generalized de Vaucouleur’s (or Sérsic) profiles.

\[
I(r) = I_e \exp \left( -b_n \left( \frac{r}{r_e} \right)^{1/n} - 1 \right)
\]

Gérard de Vaucouleurs (1918-1995) was first to show in 1948 that this function resembles the galaxy light profiles, with the only difference that he held \( n \) fixed to a value of 4. Twenty years later in 1968, J. L. Sérsic showed that \( n \) can have a variety of values and does not necessarily need to be 4. This profile depends on the effective radius \( (r_e) \) which is defined as the radius which contains half of the profile brightness (see Section 8.1.3 [Profile magnitude], page 446). \( I_e \) is the flux at the effective radius. The Sérsic index \( n \) is used to define the concentration of the profile within \( r_e \) and \( b_n \) is a constant dependent on \( n \). MacArthur et al.\(^4\) show that for \( n > 0.35 \), \( b_n \) can be accurately approximated using this equation:

\[
b_n = 2n - \frac{1}{3} + \frac{4}{405n} + \frac{46}{25515n^2} + \frac{131}{1148175n^3} - \frac{2194697}{30690717750n^4}
\]

### 8.1.1.5 Sampling from a function

A pixel is the ultimate level of accuracy to gather data, we can't get any more accurate in one image, this is known as sampling in signal processing. However, the mathematical profiles which describe our models have infinite accuracy. Over a large fraction of the area of astrophysically interesting profiles (for example galaxies or PSFs), the variation of the profile over the area of one pixel is not too significant. In such cases, the elliptical radius \( (r_e) \) of the center of the pixel can be assigned as the final value of the pixel, (see Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440).

As you approach their center, some galaxies become very sharp (their value significantly changes over one pixel’s area). This sharpness increases with smaller effective radius and larger Sérsic values. Thus rendering the central value extremely inaccurate. The first method that comes to mind for solving this problem is integration. The functional form

of the profile can be integrated over the pixel area in a 2D integration process. However, unfortunately numerical integration techniques also have their limitations and when such sharp profiles are needed they can become extremely inaccurate.

The most accurate method of sampling a continuous profile on a discrete space is by choosing a large number of random points within the boundaries of the pixel and taking their average value (or Monte Carlo integration). This is also, generally speaking, what happens in practice with the photons on the pixel. The number of random points can be set with --numrandom.

Unfortunately, repeating this Monte Carlo process would be extremely time and CPU consuming if it is to be applied to every pixel. In order to not lose too much accuracy, in MakeProfiles, the profile is built using both methods explained below. The building of the profile begins from its central pixel and continues (radially) outwards. Monte Carlo integration is first applied (which yields $F_r$), then the central pixel value ($F_c$) is calculated on the same pixel. If the fractional difference ($|F_r - F_c|/F_r$) is lower than a given tolerance level (specified with --tolerance) MakeProfiles will stop using Monte Carlo integration and only use the central pixel value.

The ordering of the pixels in this inside-out construction is based on $r = \sqrt{(i_c - i)^2 + (j_c - j)^2}$, not $r_{el}$, see Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440. When the axis ratios are large (near one) this is fine. But when they are small and the object is highly elliptical, it might seem more reasonable to follow $r_{el}$ not $r$. The problem is that the gradient is stronger in pixels with smaller $r$ (and larger $r_{el}$) than those with smaller $r_{el}$. In other words, the gradient is strongest along the minor axis. So if the next pixel is chosen based on $r_{el}$, the tolerance level will be reached sooner and lots of pixels with large fractional differences will be missed.

Monte Carlo integration uses a random number of points. Thus, every time you run it, by default, you will get a different distribution of points to sample within the pixel. In the case of large profiles, this will result in a slight difference of the pixels which use Monte Carlo integration each time MakeProfiles is run. To have a deterministic result, you have to fix the random number generator properties which is used to build the random distribution. This can be done by setting the GSL_RNG_TYPE and GSL_RNG_SEED environment variables and calling MakeProfiles with the --envseed option. To learn more about the process of generating random numbers, see Section 8.2.1.4 [Generating random numbers], page 464.

The seed values are fixed for every profile: with --envseed, all the profiles have the same seed and without it, each will get a different seed using the system clock (which is accurate to within one microsecond). The same seed will be used to generate a random number for all the sub-pixel positions of all the profiles. So in the former, the sub-pixel points checked for all the pixels undergoing Monte Carlo integration in all profiles will be identical. In other words, the sub-pixel points in the first (closest to the center) pixel of all the profiles will be identical with each other. All the second pixels studied for all the profiles will also receive an identical (different from the first pixel) set of sub-pixel points and so on. As long as the number of random points used is large enough or the profiles are not identical, this should not cause any systematic bias.

8.1.1.6 Oversampling

The steps explained in Section 8.1.1.5 [Sampling from a function], page 444, do give an accurate representation of a profile prior to convolution. However, in an actual observation,
the image is first convolved with or blurred by the atmospheric and instrument PSF in a continuous space and then it is sampled on the discrete pixels of the camera.

In order to more accurately simulate this process, the unconvolved image and the PSF are created on a finer pixel grid. In other words, the output image is a certain odd-integer multiple of the desired size, we can call this ‘oversampling’. The user can specify this multiple as a command-line option. The reason this has to be an odd number is that the PSF has to be centered on the center of its image. An image with an even number of pixels on each side does not have a central pixel.

The image can then be convolved with the PSF (which should also be oversampled on the same scale). Finally, image can be sub-sampled to get to the initial desired pixel size of the output image. After this, mock noise can be added as explained in the next section. This is because unlike the PSF, the noise occurs in each output pixel, not on a continuous space like all the prior steps.

### 8.1.2 If convolving afterwards

In case you want to convolve the image later with a given point spread function, make sure to use a larger image size. After convolution, the profiles become larger and a profile that is normally completely outside of the image might fall within it.

On one axis, if you want your final (convolved) image to be $m$ pixels and your PSF is $2n + 1$ pixels wide, then when calling MakeProfiles, set the axis size to $m + 2n$, not $m$. You also have to shift all the pixel positions of the profile centers on the that axis by $n$ pixels to the positive.

After convolution, you can crop the outer $n$ pixels with the section crop box specification of Crop: `--section=n+1:*-n,n+1:*-n` (according to the FITS standard, counting is from 1 so we use $n+1$) assuming your PSF is a square, see Section 6.1.2 [Crop section syntax], page 262. This will also remove all discrete Fourier transform artifacts (blurred sides) from the final image. To facilitate this shift, MakeProfiles has the options `--xshift`, `--yshift` and `--prepforconv`, see Section 8.1.4 [Invoking MakeProfiles], page 447.

### 8.1.3 Profile magnitude

To find the profile brightness or its magnitude, (see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394), it is customary to use the 2D integration of the flux to infinity. However, in MakeProfiles we do not follow this idealistic approach and apply a more realistic method to find the total brightness or magnitude: the sum of all the pixels belonging to a profile within its predefined truncation radius. Note that if the truncation radius is not large enough, this can be significantly different from the total integrated light to infinity.

An integration to infinity is not a realistic condition because no galaxy extends indefinitely (important for high Sérsic index profiles), pixelation can also cause a significant difference between the actual total pixel sum value of the profile and that of integration to infinity, especially in small and high Sérsic index profiles. To be safe, you can specify a large enough truncation radius for such compact high Sérsic index profiles.

If oversampling is used then the brightness is calculated using the over-sampled image, see Section 8.1.1.6 [Oversampling], page 445, which is much more accurate. The profile is first built in an array completely bounding it with a normalization constant of unity (see
Section 8.1.1.4 [Galaxies], page 444. Taking $B$ to be the desired brightness and $S$ to be the sum of the pixels in the created profile, every pixel is then multiplied by $B/S$ so the sum is exactly $B$.

If the --individual option is called, this same array is written to a FITS file. If not, only the overlapping pixels of this array and the output image are kept and added to the output array.

8.1.4 Invoking MakeProfiles

MakeProfiles will make any number of profiles specified in a catalog either individually or in one image. The executable name is astmkprof with the following general template

$$\texttt{astmkprof [OPTION ...] [Catalog]}$$

One line examples:

## Make an image with profiles in catalog.txt (with default size):
$$\texttt{astmkprof catalog.txt}$$

## Make the profiles in catalog.txt over image.fits:
$$\texttt{astmkprof --background=image.fits catalog.txt}$$

## Make a Moffat PSF with FWHM 3pix, beta=2.8, truncation=5
$$\texttt{astmkprof --kernel=moffat,3,2.8,5 --oversample=1}$$

## Make profiles in catalog, using RA and Dec in the given column:
$$\texttt{astmkprof --ccol=RA_CENTER --ccol=DEC_CENTER --mode=wcs catalog.txt}$$

## Make a 1500x1500 merged image (oversampled 500x500) image along
## with an individual image for all the profiles in catalog:
$$\texttt{astmkprof --individual --oversample 3 --mergedsize=500,500 cat.txt}$$

The parameters of the mock profiles can either be given through a catalog (which stores the parameters of many mock profiles, see Section 8.1.4.1 [MakeProfiles catalog], page 448), or the --kernel option (see Section 8.1.4.3 [MakeProfiles output dataset], page 457). The catalog can be in the FITS ASCII, FITS binary format, or plain text formats (see Section 4.7 [Tables], page 183). A plain text catalog can also be provided using the Standard input (see Section 4.1.4 [Standard input], page 167). The columns related to each parameter can be determined both by number, or by match/search criteria using the column names, units, or comments, with the options ending in col, see below.

Without any file given to the --background option, MakeProfiles will make a zero-valued image and build the profiles on that (its size and main WCS parameters can also be defined through the options described in Section 8.1.4.3 [MakeProfiles output dataset], page 457). Besides the main/merged image containing all the profiles in the catalog, it is also possible to build individual images for each profile (only enclosing one full profile to its truncation radius) with the --individual option.

If an image is given to the --background option, the pixels of that image are used as the background value for every pixel hence flux value of each profile pixel will be added to the pixel in that background value. You can disable this with the --clearcanvas option (which will initialize the background to zero-valued pixels and build profiles over that).
With the `--background` option, the values to all options relating to the “canvas” (output size and WCS) will be ignored if specified, for example `--oversample`, `--mergedsize`, and `--prepforconv`.

The sections below discuss the options specific to MakeProfiles based on context: the input catalog settings which can have many rows for different profiles are discussed in Section 8.1.4.1 [MakeProfiles catalog], page 448, in Section 8.1.4.2 [MakeProfiles profile settings], page 451, we discuss how you can set general profile settings (that are the same for all the profiles in the catalog). Finally Section 8.1.4.3 [MakeProfiles output dataset], page 457, and Section 8.1.4.4 [MakeProfiles log file], page 461, discuss the outputs of MakeProfiles and how you can configure them. Besides these, MakeProfiles also supports all the common Gnuastro program options that are discussed in Section 4.1.2 [Common options], page 156, so please flip through them is well for a more comfortable usage.

When building 3D profiles, there are more degrees of freedom. Hence, more columns are necessary and all the values related to dimensions (for example size of dataset in each dimension and the WCS properties) must also have 3 values. To allow having an independent set of default values for creating 3D profiles, MakeProfiles also installs a `astmkprof-3d.conf` configuration file (see Section 4.2 [Configuration files], page 169). You can use this for default 3D profile values. For example, if you installed Gnuastro with the prefix `/usr/local` (the default location, see Section 3.3.1.2 [Installation directory], page 139), you can benefit from this configuration file by running MakeProfiles like the example below. As with all configuration files, if you want to customize a given option, call it before the configuration file.

```
$ astmkprof --config=/usr/local/etc/astmkprof-3d.conf catalog.txt
```

To further simplify the process, you can define a shell alias in any startup file (for example `~/.bashrc`, see Section 3.3.1.2 [Installation directory], page 139). Assuming that you installed Gnuastro in `/usr/local`, you can add this line to the startup file (you may put it all in one line, it is broken into two lines here for fitting within page limits).

```
alias astmkprof-3d="astmkprof --config=/usr/local/etc/astmkprof-3d.conf"
```

Using this alias, you can call MakeProfiles with the name `astmkprof-3d` (instead of `astmkprof`). It will automatically load the 3D specific configuration file first, and then parse any other arguments, options or configuration files. You can change the default values in this 3D configuration file by calling them on the command-line as you do with `astmkprof`.

Please see Section 2.1 [Sufi simulates a detection], page 18, for a very complete tutorial explaining how one could use MakeProfiles in conjunction with other Gnuastro’s programs to make a complete simulated image of a mock galaxy.

### 8.1.4.1 MakeProfiles catalog

The catalog containing information about each profile can be in the FITS ASCII, FITS binary, or plain text formats (see Section 4.7 [Tables], page 183). The latter can also be provided using standard input (see Section 4.1.4 [Standard input], page 167). Its columns can be ordered in any desired manner. You can specify which columns belong to which

---

5 Recall that for single-invocation options, the last command-line invocation takes precedence over all previous invocations (including those in the 3D configuration file). See the description of `--config` in Section 4.1.2.3 [Operating mode options], page 162.
parameters using the set of options discussed below. For example through the --rcol and --tcol options, you can specify the column that contains the radial parameter for each profile and its truncation respectively. See Section 4.7.3 [Selecting table columns], page 188, for a thorough discussion on the values to these options.

The value for the profile center in the catalog (the --ccol option) can be a floating point number so the profile center can be on any sub-pixel position. Note that pixel positions in the FITS standard start from 1 and an integer is the pixel center. So a 2D image actually starts from the position (0.5, 0.5), which is the bottom-left corner of the first pixel. When a --background image with WCS information is provided or you specify the WCS parameters with the respective options, you may also use RA and Dec to identify the center of each profile (see the --mode option below).

In MakeProfiles, profile centers do not have to be in (overlap with) the final image. Even if only one pixel of the profile within the truncation radius overlaps with the final image size, the profile is built and included in the final image. Profiles that are completely out of the image will not be created (unless you explicitly ask for it with the --individual option). You can use the output log file (created with --log to see which profiles were within the image, see Section 4.1.2 [Common options], page 156).

If PSF profiles (Moffat or Gaussian, see Section 8.1.1.2 [Point spread function], page 442) are in the catalog and the profiles are to be built in one image (when --individual is not used), it is assumed they are the PSF(s) you want to convolve your created image with. So by default, they will not be built in the output image but as separate files. The sum of pixels of these separate files will also be set to unity (1) so you are ready to convolve, see Section 6.3.1.1 [Convolution process], page 307. As a summary, the position and magnitude of PSF profile will be ignored. This behavior can be disabled with the --psfinimg option. If you want to create all the profiles separately (with --individual) and you want the sum of the PSF profile pixels to be unity, you have to set their magnitudes in the catalog to the zero point magnitude and be sure that the central positions of the profiles don’t have any fractional part (the PSF center has to be in the center of the pixel).

The list of options directly related to the input catalog columns is shown below.

--ccol=STR/INT
Center coordinate column for each dimension. This option must be called two times to define the center coordinates in an image. For example --ccol=RA and --ccol=DEC (along with --mode=wcs) will inform MakeProfiles to look into the catalog columns named RA and DEC for the Right Ascension and Declination of the profile centers.

--fcol=INT/STR
The functional form of the profile with one of the values below depending on the desired profile. The column can contain either the numeric codes (for example ‘1’) or string characters (for example ‘sersic’). The numeric codes are easier to use in scripts which generate catalogs with hundreds or thousands of profiles. The string format can be easier when the catalog is to be written/checked by hand/eye before running MakeProfiles. It is much more readable and provides a level of documentation. All Gnuastro’s recognized table formats (see Section 4.7.1 [Recognized table formats], page 184) accept string type columns.
To have string columns in a plain text table/catalog, see Section 4.7.2 [Gnuastro text table format], page 185.

- Sérsic profile with ‘sersic’ or ‘1’.
- Moffat profile with ‘moffat’ or ‘2’.
- Gaussian profile with ‘gaussian’ or ‘3’.
- Point source with ‘point’ or ‘4’.
- Flat profile with ‘flat’ or ‘5’.
- Circumference profile with ‘circum’ or ‘6’. A fixed value will be used for all pixels less than or equal to the truncation radius ($r_t$) and greater than $r_t - w$ ($w$ is the value to the --circumwidth).
- Radial distance profile with ‘distance’ or ‘7’. At the lowest level, each pixel only has an elliptical radial distance given the profile’s shape and orientation (see Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440). When this profile is chosen, the pixel’s elliptical radial distance from the profile center is written as its value. For this profile, the value in the magnitude column (--mcol) will be ignored. You can use this for checks or as a first approximation to define your own higher-level radial function. In the latter case, just note that the central values are going to be incorrect (see Section 8.1.1.5 [Sampling from a function], page 444).
- Custom radial profile with ‘custom-prof’ or ‘8’. The values to use for each radial interval should be in the table given to --customtable. By default, once the profile is built with the given values, it will be scaled to have a total magnitude that you have requested in the magnitude column of the profile (in --mcol). If you want the raw values in the 2D profile (to ignore the magnitude column), use --mcolnocustprof. For more, see the description of --customtable in Section 8.1.4.2 [MakeProfiles profile settings], page 451.
- Azimuthal angle profile with ‘azimuth’ or ‘9’. Every pixel within the truncation radius will be given its azimuthal angle (in degrees, from 0 to 360) from the major axis. In combination with the radial distance profile, you can now create complex features in polar coordinates, such as tidal tails or tidal shocks (using the Arithmetic program to mix the radius and azimuthal angle through a function to create your desired features).
- Custom image with ‘custom-img’ or ‘10’. The image(s) to use should be given to the --customimg option (which can be called multiple times for multiple images). To identify which one of the images (given to --customimg) should be used, you should specify their counter in the “radius” column below. For more, see the description of custom-img in Section 8.1.4.2 [MakeProfiles profile settings], page 451.

```
--rcol=STR/INT
```

The radius parameter of the profiles. Effective radius ($r_e$) if Sérsic, FWHM if Moffat or Gaussian.
For a custom image profile, this option is not interpreted as a radius, but as a counter (identifying which one of the images given to --customimg should be used for each row).

--ncol=STR/INT
The Sérsic index \( n \) or Moffat \( \beta \).

--pcol=STR/INT
The position angle (in degrees) of the profiles relative to the first FITS axis (horizontal when viewed in SAO DS9). When building a 3D profile, this is the first Euler angle: first rotation of the ellipsoid major axis from the first FITS axis (rotating about the third axis). See Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440.

--p2col=STR/INT
Second Euler angle (in degrees) when building a 3D ellipsoid. This is the second rotation of the ellipsoid major axis (following --pcol) about the (rotated) X axis. See Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440. This column is ignored when building a 2D profile.

--p3col=STR/INT
Third Euler angle (in degrees) when building a 3D ellipsoid. This is the third rotation of the ellipsoid major axis (following --pcol and --p2col) about the (rotated) Z axis. See Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440. This column is ignored when building a 2D profile.

--qcol=STR/INT
The axis ratio of the profiles (minor axis divided by the major axis in a 2D ellipse). When building a 3D ellipse, this is the ratio of the major axis to the semi-axis length of the second dimension (in a right-handed coordinate system). See \( q_1 \) in Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440.

--q2col=STR/INT
The ratio of the ellipsoid major axis to the third semi-axis length (in a right-handed coordinate system) of a 3D ellipsoid. See \( q_1 \) in Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440. This column is ignored when building a 2D profile.

--mcol=STR/INT
The total pixelated magnitude of the profile within the truncation radius, see Section 8.1.3 [Profile magnitude], page 446.

--tcol=STR/INT
The truncation radius of this profile. By default it is in units of the radial parameter of the profile (the value in the --rcol of the catalog). If --tunitinp is given, this value is interpreted in units of pixels (prior to oversampling) irrespective of the profile.

8.1.4.2 MakeProfiles profile settings
The profile parameters that differ between each created profile are specified through the columns in the input catalog and described in Section 8.1.4.1 [MakeProfiles catalog],
Besides those there are general settings for some profiles that don’t differ between one profile and another, they are a property of the general process. For example how many random points to use in the monte-carlo integration, this value is fixed for all the profiles. The options described in this section are for configuring such properties.

```
--mode=STR
```
Interpret the center position columns (``--ccol`` in Section 8.1.4.1 [MakeProfiles catalog], page 448) in image or WCS coordinates. This option thus accepts only two values: `img` and `wcs`. It is mandatory when a catalog is being used as input.

```
-r
```
```
--numrandom
```
The number of random points used in the central regions of the profile, see Section 8.1.1.5 [Sampling from a function], page 444.

```
-e
```
```
--envseed
```
Use the value to the `GSL_RNG_SEED` environment variable to generate the random Monte Carlo sampling distribution, see Section 8.1.1.5 [Sampling from a function], page 444, and Section 8.2.1.4 [Generating random numbers], page 464.

```
-t FLT
```
```
--tolerance=FLT
```
The tolerance to switch from Monte Carlo integration to the central pixel value, see Section 8.1.1.5 [Sampling from a function], page 444.

```
-p
```
```
--tunitinp
```
The truncation column of the catalog is in units of pixels. By default, the truncation column is considered to be in units of the radial parameters of the profile (``--rcol``). Read it as ‘t-unit-in-p’ for ‘truncation unit in pixels’.

```
-f
```
```
--mforflatpix
```
When making fixed value profiles (flat and circumference, see ‘``--fcol``’), don’t use the value in the column specified by ‘``--mcol``’ as the magnitude. Instead use it as the exact value that all the pixels of these profiles should have. This option is irrelevant for other types of profiles. This option is very useful for creating masks, or labeled regions in an image. Any integer, or floating point value can used in this column with this option, including `NaN` (or ‘nan’, or ‘NAN’, case is irrelevant), and infinities (``inf``, ``-inf``, or ``+inf``).

For example, with this option if you set the value in the magnitude column (``--mcol``) to `NaN`, you can create an elliptical or circular mask over an image (which can be given as the argument), see Section 6.1.3 [Blank pixels], page 262. Another useful application of this option is to create labeled elliptical or circular apertures in an image. To do this, set the value in the magnitude column to the label you want for this profile. This labeled image can then be used in combination with NoiseChisel’s output (see Section 7.2.2.3 [NoiseChisel output], page 378) to do aperture photometry with MakeCatalog (see Section 7.4 [MakeCatalog], page 392).
Alternatively, if you want to mark regions of the image (for example with an elliptical circumference) and you don’t want to use NaN values (as explained above) for some technical reason, you can get the minimum or maximum value in the image\(^6\) using Arithmetic (see Section 6.2 [Arithmetic], page 272), then use that value in the magnitude column along with this option for all the profiles.

Please note that when using MakeProfiles on an already existing image, you have to set ‘\(--\text{oversample}=1\)’. Otherwise all the profiles will be scaled up based on the oversampling scale in your configuration files (see Section 4.2 [Configuration files], page 169) unless you have accounted for oversampling in your catalog.

\textbf{--mcolisbrightness}

The value given in the “magnitude column” (specified by \texttt{--mcol}, see Section 8.1.4.1 [MakeProfiles catalog], page 448) must be interpreted as brightness, not magnitude. The zero point magnitude (value to the \texttt{--zeropoint} option) is ignored and the given value must have the same units as the input dataset’s pixels.

Recall that the total profile magnitude or brightness that is specified with in the \texttt{--mcol} column of the input catalog is not an integration to infinity, but the actual sum of pixels in the profile (until the desired truncation radius). See Section 8.1.3 [Profile magnitude], page 446, for more on this point.

\textbf{--mcolnocustprof}

Don’t touch (re-scale) the custom profile that should be inserted in \texttt{custom-prof} profile (see the description of \texttt{--fcol} in Section 8.1.4.1 [MakeProfiles catalog], page 448, or the description of \texttt{--customtable} below). By default, MakeProfiles will scale (multiply) the custom image’s pixels to have the desired magnitude (or brightness if \texttt{--mcolisbrightness} is called) in that row.

\textbf{--mcolnocustimg}

Don’t touch (re-scale) the custom image that should be inserted in \texttt{custom-img} profile (see the description of \texttt{--fcol} in Section 8.1.4.1 [MakeProfiles catalog], page 448). By default, MakeProfiles will scale (multiply) the custom image’s pixels to have the desired magnitude (or brightness if \texttt{--mcolisbrightness} is called) in that row.

\textbf{--magatpeak}

The magnitude column in the catalog (see Section 8.1.4.1 [MakeProfiles catalog], page 448) will be used to set the brightness only for the profile’s peak (maximum) pixel, not the full profile. Note that this is the flux of the profile’s peak (maximum) pixel in the final output of MakeProfiles. So beware of the oversampling, see Section 8.1.1.6 [Oversampling], page 445.

This option can be useful if you want to check a mock profile’s total magnitude at various truncation radii. Without this option, no matter what the truncation radius is, the total magnitude will be the same as that given in the catalog. But

\footnote{\textit{The minimum will give a better result, because the maximum can be too high compared to most pixels in the image, making it harder to display.}}
with this option, the total magnitude will become brighter as you increase the truncation radius.

In sharper profiles, sometimes the accuracy of measuring the peak profile flux is more than the overall object brightness. In such cases, with this option, the final profile will be built such that its peak has the given magnitude, not the total profile.

**CAUTION:** If you want to use this option for comparing with observations, please note that MakeProfiles does not do convolution. Unless you have de-convolved your data, your images are convolved with the instrument and atmospheric PSF, see Section 8.1.1.2 [Point spread function], page 442. Particularly in sharper profiles, the flux in the peak pixel is strongly decreased after convolution. Also note that in such cases, besides de-convolution, you will have to set **--oversample=1** otherwise after resampling your profile with Warp (see Section 6.4 [Warp], page 328), the peak flux will be different.

---

**--customtable** FITS/TXT

The filename of the table to use in the custom radial profiles (see description of **--fcol** in Section 8.1.4.1 [MakeProfiles catalog], page 448. This can be a plain-text table, or FITS table, see Section 4.7 [Tables], page 183, if it is a FITS table, you can use **--customtablehdu** to specify which HDU should be used (described below).

A custom radial profile can have any value you want for a given radial profile (including NaN/blank values). Each interval is defined by its minimum (inclusive) and maximum (exclusive) radius, when a pixel center falls within a radius interval, the value specified for that interval will be used. If a pixel is not in the given intervals, a value of 0.0 will be used for that pixel.

The table should have 3 columns as shown below. If the intervals are contiguous (the maximum value of the previous interval is equal to the minimum value of an interval) and the intervals all have the same size (difference between minimum and maximum values) the creation of these profiles will be fast. However, if the intervals are not sorted and contiguous, Makeprofiles will parse the intervals from the top of the table and use the first interval that contains the pixel center.

**Column 1:** The interval’s minimum radius.

**Column 2:** The interval’s maximum radius.

**Column 3:** The value to be used for pixels within the given interval (including NaN/blank).

For example let’s assume you have the radial profile below in a file called *radial.txt*. The first column is the larger interval radius (in units of pixels) and the second column is the value in that interval:

```
1  100
2   90
3   50
4    10
```
You can construct the table to give to `--customtable` with the command below (using Gnuastro’s Section 5.3.1 [Column arithmetic], page 230).

```
asttable radial.txt -c'arith $1 1 -' -c1,2 -ocustom.fits
```

In case the intervals are different from 1 (for example 0.5), change the `$1 1 -` to `$1 0.5 -`. On a side-note, Gnuastro has features to extract the radial profile of an object from the image, see Section 10.2 [Generate radial profile], page 486.

By default, once a 2D image is constructed for the radial profile, it will be scaled such that its total magnitude corresponds to the value in the magnitude column (``--mcol``) of the main input catalog. If you want to disable the scaling and use the raw values in your custom profile (in other words: you want to ignore the magnitude column) you need to call `--mcolnocustprof` (see above).

```
--customtablehdud INT/STR
The HDU/extension in the FITS file given to `--customtable`.

--customimg=STR[,STR]
A custom FITS image that should be used for the `custom-img` profiles (see the description of `--fcol` in Section 8.1.4.1 [MakeProfiles catalog], page 448). Multiple files can be given to this option (separated by a comma), and this option can be called multiple times itself (useful when many custom image profiles should be added). If the HDU of the images are different, you can use `--customimghdu` (described below).

Through the “radius” column, MakeProfiles will know which one of the images given to this option should be used in each row. For example let’s assume your input catalog (``cat.fits``) has the following contents (output of first command below), and you call MakeProfiles like the second command below to insert four profiles into the background `back.fits` image.

The first profile below is Sersic (with an `--fcol`, or 4-th column, code of 1). So MakeProfiles builds the pixels of the first profile, and all column values are meaningful. However, the second, third and fourth inserted objects are custom images (with an `--fcol` code of 10). For the custom image profiles, you see that the radius column has values of 1 or 2. This tells MakeProfiles to use the first image given to `--customimg` (or `gal-1.fits`) for the second and fourth inserted objects. The second image given to `--customimg` (or `gal-2.fits`) will be used for the third inserted object. Finally, all three custom image profiles have different magnitudes, and the values in `--ncol`, `--pcol`, `--qcol` and `--tcol` are ignored.

```
$ cat cat.fits
1 53.15506 -27.785165 1 20 1 20 0.6 25 5
2 53.15602 -27.777887 10 1 0 0 0 22 0
3 53.16440 -27.775876 10 2 0 0 0 24 0
4 53.16849 -27.787406 10 1 0 0 0 23 0

$ astmkprof cat.fits --mode=wcs --zeropoint=25.68 \
Chapter 8: Modeling and fitting

--background=back.fits --output=out.fits \
--customim=gal-1.fits --customim=gal-2.fits

--customimghdu=INT/STR
The HDU(s) of the images given to --customimghdu. If this option is only called once, but --customim is called many times, MakeProfiles will assume that all images given to --customim have the same HDU. Otherwise (if the number of HDUs is equal to the number of images), then each image will use its corresponding HDU.

-X INT,INT
--shift=INT,INT
Shift all the profiles and enlarge the image along each dimension. To better understand this option, please see n in Section 8.1.2 [If convolving afterwards], page 446. This is useful when you want to convolve the image afterwards. If you are using an external PSF, be sure to oversample it to the same scale used for creating the mock images. If a background image is specified, any possible value to this option is ignored.

-c
--prepforconv
Shift all the profiles and enlarge the image based on half the width of the first Moffat or Gaussian profile in the catalog, considering any possible oversampling see Section 8.1.2 [If convolving afterwards], page 446. --prepforconv is only checked and possibly activated if --xshift and --yshift are both zero (after reading the command-line and configuration files). If a background image is specified, any possible value to this option is ignored.

-z FLT
--zeropoint=FLT
The zero point magnitude of the input. For more on the zero point magnitude, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

-w FLT
--circumwidth=FLT
The width of the circumference if the profile is to be an elliptical circumference or annulus. See the explanations for this type of profile in --fcol.

-R
--replace
Do not add the pixels of each profile over the background, or other profiles. But replace the values.
By default, when two profiles overlap, the final pixel value is the sum of all the profiles that overlap on that pixel. This is the expected situation when dealing with physical object profiles like galaxies or stars/PSF. However, when MakeProfiles is used to build integer labeled images (for example in Section 2.2.17 [Aperture photometry], page 65), this is not the expected situation: the sum of two labels will be a new label. With this option, the pixels are not added but the largest (maximum) value over that pixel is used. Because the maximum operator is independent of the order of values, the output is also thread-safe.
8.1.4.3 MakeProfiles output dataset

MakeProfiles takes an input catalog uses basic properties that are defined there to build a dataset, for example a 2D image containing the profiles in the catalog. In Section 8.1.4.1 [MakeProfiles catalog], page 448, and Section 8.1.4.2 [MakeProfiles profile settings], page 451, the catalog and profile settings were discussed. The options of this section, allow you to configure the output dataset (or the canvas that will host the built profiles).

-k FITS
--background=FITS

A background image FITS file to build the profiles on. The extension that contains the image should be specified with the --backhdu option, see below. When a background image is specified, it will be used to derive all the information about the output image. Hence, the following options will be ignored: --mergesize, --oversample, --crpix, --crval (generally, all other WCS related parameters) and the output’s data type (see --type in Section 4.1.2.1 [Input/Output options], page 157).

The image will act like a canvas to build the profiles on: profile pixel values will be summed with the background image pixel values. With the --replace option you can disable this behavior and replace the profile pixels with the background pixels. If you want to use all the image information above, except for the pixel values (you want to have a blank canvas to build the profiles on, based on an input image), you can call --clearcanvas, to set all the input image’s pixels to zero before starting to build the profiles over it (this is done in memory after reading the input, so nothing will happen to your input file).

-B STR/INT
--backhdu=STR/INT

The header data unit (HDU) of the file given to --background.

-C
--clearcanvas

When an input image is specified (with the --background option, set all its pixels to 0.0 immediately after reading it into memory. Effectively, this will allow you to use all its properties (described under the --background option), without having to worry about the pixel values.

--clearcanvas can come in handy in many situations, for example if you want to create a labeled image (segmentation map) for creating a catalog (see Section 7.4 [MakeCatalog], page 392). In other cases, you might have modeled the objects in an image and want to create them on the same frame, but without the original pixel values.

-E STR/INT,FLT[,FLT,[...]]
--kernel=STR/INT,FLT[,FLT,[...]]

Only build one kernel profile with the parameters given as the values to this option. The different values must be separated by a comma (,). The first value identifies the radial function of the profile, either through a string or through a number (see description of --fcol in Section 8.1.4.1 [MakeProfiles catalog], page 448). Each radial profile needs a different total number of parameters:
Sérsic and Moffat functions need 3 parameters: radial, Sérsic index or Moffat β, and truncation radius. The Gaussian function needs two parameters: radial and truncation radius. The point function doesn’t need any parameters and flat and circumference profiles just need one parameter (truncation radius).

The PSF or kernel is a unique (and highly constrained) type of profile: the sum of its pixels must be one, its center must be the center of the central pixel (in an image with an odd number of pixels on each side), and commonly it is circular, so its axis ratio and position angle are one and zero respectively. Kernels are commonly necessary for various data analysis and data manipulation steps (for example see Section 6.3 [Convolve], page 306, and Section 7.2 [NoiseChisel], page 362. Because of this it is inconvenient to define a catalog with one row and many zero valued columns (for all the non-necessary parameters). Hence, with this option, it is possible to create a kernel with MakeProfiles without the need to create a catalog. Here are some examples:

```
--kernel=moffat,3,2.8,5
  A Moffat kernel with FWHM of 3 pixels, β = 2.8 which is truncated at 5 times the FWHM.

--kernel=gaussian,2,3
  A circular Gaussian kernel with FWHM of 2 pixels and truncated at 3 times the FWHM.
```

This option may also be used to create a 3D kernel. To do that, two small modifications are necessary: add a `-3d` (or `-3D`) to the profile name (for example `moffat-3d`) and add a number (axis-ratio along the third dimension) to the end of the parameters for all profiles except `point`. The main reason behind providing an axis ratio in the third dimension is that in 3D astronomical datasets, commonly the third dimension doesn’t have the same nature (units/sampling) as the first and second.

For example in IFU (optical) or Radio datacubes, the first and second dimensions are commonly spatial/angular positions (like RA and Dec) but the third dimension is wavelength or frequency (in units of Angstroms for Herz). Because of this different nature (which also affects the processing), it may be necessary for the kernel to have a different extent in that direction.

If the 3rd dimension axis ratio is equal to 1.0, then the kernel will be a spheroid. If its smaller than 1.0, the kernel will be button-shaped: extended less in the third dimension. However, when it is larger than 1.0, the kernel will be bullet-shaped: extended more in the third dimension. In the latter case, the radial parameter will correspond to the length along the 3rd dimension. For example, let’s have a look at the two examples above but in 3D:

```
--kernel=moffat-3d,3,2.8,5,0.5
  An ellipsoid Moffat kernel with FWHM of 3 pixels, β = 2.8 which is truncated at 5 times the FWHM. The ellipsoid is circular in the first two dimensions, but in the third dimension its extent is half the first two.
```
--kernel=gaussian-3d,2,3,1
A spherical Gaussian kernel with FWHM of 2 pixels and truncated at 3 times the FWHM.

Of course, if a specific kernel is needed that doesn’t fit the constraints imposed by this option, you can always use a catalog to define any arbitrary kernel. Just call the --individual and --nomerged options to make sure that it is built as a separate file (individually) and no “merged” image of the input profiles is created.

-x INT,INT
--mergedsize=INT,INT
The number of pixels along each axis of the output, in FITS order. This is before over-sampling. For example if you call MakeProfiles with --mergedsize=100,150 --oversample=5 (assuming no shift due for later convolution), then the final image size along the first axis will be 500 by 750 pixels. Fractions are acceptable as values for each dimension, however, they must reduce to an integer, so --mergedsize=150/3,300/3 is acceptable but --mergedsize=150/4,300/4 is not.

When viewing a FITS image in DS9, the first FITS dimension is in the horizontal direction and the second is vertical. As an example, the image created with the example above will have 500 pixels horizontally and 750 pixels vertically.

If a background image is specified, this option is ignored.

-s INT
--oversample=INT
The scale to over-sample the profiles and final image. If not an odd number, will be added by one, see Section 8.1.1.6 [Oversampling], page 445. Note that this --oversample will remain active even if an input image is specified. If your input catalog is based on the background image, be sure to set --oversample=1.

--psfinimg
Build the possibly existing PSF profiles (Moffat or Gaussian) in the catalog into the final image. By default they are built separately so you can convolve your images with them, thus their magnitude and positions are ignored. With this option, they will be built in the final image like every other galaxy profile. To have a final PSF in your image, make a point profile where you want the PSF and after convolution it will be the PSF.

-i
--individual
If this option is called, each profile is created in a separate FITS file within the same directory as the output and the row number of the profile (starting from zero) in the name. The file for each row’s profile will be in the same directory as the final combined image of all the profiles and will have the final image’s name as a suffix. So for example if the final combined image is named ./out/fromcatalog.fits, then the first profile that will be created with this option will be named ./out/0_fromcatalog.fits.

Since each image only has one full profile out to the truncation radius the profile is centered and so, only the sub-pixel position of the profile center is important.
for the outputs of this option. The output will have an odd number of pixels. If there is no oversampling, the central pixel will contain the profile center. If the value to --oversample is larger than unity, then the profile center is on any of the central --oversample’d pixels depending on the fractional value of the profile center.

If the fractional value is larger than half, it is on the bottom half of the central region. This is due to the FITS definition of a real number position: The center of a pixel has fractional value 0.00 so each pixel contains these fractions: .5 – 0.75 – 0.00 (pixel center) – 0.25 – .5.

--m
--nomerged

Don’t make a merged image. By default after making the profiles, they are added to a final image with side lengths specified by --mergedsize if they overlap with it.

The options below can be used to define the world coordinate system (WCS) properties of the MakeProfiles outputs. The option names are deliberately chosen to be the same as the FITS standard WCS keywords. See Section 8 of Pence et al [2010] (https://doi.org/10.1051/0004-6361/201015362) for a short introduction to WCS in the FITS standard

If you look into the headers of a FITS image with WCS for example you will see all these names but in uppercase and with numbers to represent the dimensions, for example CRPIX1 and PC2_1. You can see the FITS headers with Gnuastro’s program using a command like this: $ astfits -p image.fits.

If the values given to any of these options does not correspond to the number of dimensions in the output dataset, then no WCS information will be added.

--crpix=FLT,FLT

The pixel coordinates of the WCS reference point. Fractions are acceptable for the values of this option.

--crval=FLT,FLT

The WCS coordinates of the Reference point. Fractions are acceptable for the values of this option. The comma-separated values can either be in degrees (a single number), or sexagesimal (_h_m_ for RA, _d_m_ for Dec, or _:__: for both). In any case, the final value that will be written in the CRVAL keyword will be a floating point number in degrees (according to the FITS standard).

--cdelt=FLT,FLT

The resolution (size of one data-unit or pixel in WCS units) of the non-oversampled dataset. Fractions are acceptable for the values of this option.

---

7 The world coordinate standard in FITS is a very beautiful and powerful concept to link/associate datasets with the outside world (other datasets). The description in the FITS standard (link above) only touches the tip of the ice-burg. To learn more please see Greisen and Calabretta [2002] (https://doi.org/10.1051/0004-6361:20021326), Calabretta and Greisen [2002] (https://doi.org/10.1051/0004-6361:20021327), Greisen et al. [2006] (https://doi.org/10.1051/0004-6361:20053818), and Calabretta et al. (http://www.atnf.csiro.au/people/mcalabre/WCS/dcs_20040422.pdf)
--pc=FLT,FLT,FLT,FLT
The PC matrix of the WCS rotation, see the FITS standard (link above) to better understand the PC matrix.

--cunit=STR,STR
The units of each WCS axis, for example deg. Note that these values are part of the FITS standard (link above). MakeProfiles won’t complain if you use non-standard values, but later usage of them might cause trouble.

--ctype=STR,STR
The type of each WCS axis, for example RA---TAN and DEC--TAN. Note that these values are part of the FITS standard (link above). MakeProfiles won’t complain if you use non-standard values, but later usage of them might cause trouble.

8.1.4.4 MakeProfiles log file
Besides the final merged dataset of all the profiles, or the individual datasets (see Section 8.1.4.3 [MakeProfiles output dataset], page 457), if the --log option is called MakeProfiles will also create a log file in the current directory (where you run MockProfiles). See Section 4.1.2 [Common options], page 156, for a full description of --log and other options that are shared between all Gnuastro programs. The values for each column are explained in the first few commented lines of the log file (starting with # character). Here is a more complete description.

- An ID (row number of profile in input catalog).
- The total magnitude of the profile in the output dataset. When the profile does not completely overlap with the output dataset, this will be different from your input magnitude.
- The number of pixels (in the oversampled image) which used Monte Carlo integration and not the central pixel value, see Section 8.1.1.5 [Sampling from a function], page 444.
- The fraction of flux in the Monte Carlo integrated pixels.
- If an individual image was created, this column will have a value of 1, otherwise it will have a value of 0.

8.2 MakeNoise
Real data are always buried in noise, therefore to finalize a simulation of real data (for example to test our observational algorithms) it is essential to add noise to the mock profiles created with MakeProfiles, see Section 8.1 [MakeProfiles], page 440. Below, the general principles and concepts to help understand how noise is quantified is discussed. MakeNoise options and argument are then discussed in Section 8.2.2 [Invoking MakeNoise], page 466.

8.2.1 Noise basics
Deep astronomical images, like those used in extragalactic studies, seriously suffer from noise in the data. Generally speaking, the sources of noise in an astronomical image are photon counting noise and Instrumental noise which are discussed in Section 8.2.1.1 [Photon counting noise], page 462, and Section 8.2.1.2 [Instrumental noise], page 464. This review finishes with Section 8.2.1.4 [Generating random numbers], page 464, which is a short
introduction on how random numbers are generated. We will see that while software random number generators are not perfect, they allow us to obtain a reproducible series of random numbers through setting the random number generator function and seed value. Therefore in this section, we’ll also discuss how you can set these two parameters in Gnuastro’s programs (including MakeNoise).

8.2.1.1 Photon counting noise

With the very accurate electronics used in today’s detectors, photon counting noise\(^8\) is the most significant source of uncertainty in most datasets. To understand this noise (error in counting) and its effect on the images of astronomical targets, let’s start by reviewing how a distribution produced by counting can be modeled as a parametric function.

Counting is an inherently discrete operation, which can only produce positive integer outputs (including zero). For example we can’t count 3.2 or \(-2\) of anything. We only count 0, 1, 2, 3 and so on. The distribution of values, as a result of counting efforts is formally known as the Poisson distribution (https://en.wikipedia.org/wiki/Poisson_distribution). It is associated to Siméon Denis Poisson, because he discussed it while working on the number of wrongful convictions in court cases in his 1837 book\(^9\).

Let’s take \(\lambda\) to represent the expected mean count of something. Furthermore, let’s take \(k\) to represent the output of a counting attempt (hence \(k\) is a positive integer). The probability density function of getting \(k\) counts (in each attempt, given the expected/mean count of \(\lambda\)) can be written as:

\[
f(k) = \frac{\lambda^k}{k!} e^{-\lambda}, \quad k \in \{0, 1, 2, 3, \ldots\}
\]

Because the Poisson distribution is only applicable to positive integer values (note the factorial operator, which only applies to non-negative integers), naturally it is very skewed when \(\lambda\) is near zero. One qualitative way to understand this behavior is that for smaller values near zero, there simply aren’t enough integers smaller than the mean, than integers that are larger. Therefore to accommodate all possibilities/counts, it has to be strongly skewed to the positive when the mean is small. For more on Skewness, see Section 2.3.3 [Skewness caused by signal and its measurement], page 83.

As \(\lambda\) becomes larger, the distribution becomes more and more symmetric, and the variance of that distribution is equal to its mean. In other words, the standard deviation is the square root of the mean. It can also be proved that when the mean is large, say \(\lambda > 1000\), the Poisson distribution approaches the Normal (Gaussian) distribution (https://en.wikipedia.org/wiki/Normal_distribution) with mean \(\mu = \lambda\) and standard deviation \(\sigma = \sqrt{\lambda}\). In other words, a Poisson distribution (with a sufficiently large \(\lambda\)) is simply a Gaussian that has one free parameter (\(\mu = \lambda\) and \(\sigma = \sqrt{\lambda}\)), instead of the two parameters that the Gaussian distribution originally has (independent \(\mu\) and \(\sigma\)).

In real situations, the photons/flux from our targets are combined with photons from a certain background (observationally, the Sky value). The Sky value is defined to be the

---

\(^8\) In practice, we are actually counting the electrons that are produced by each photon, not the actual photons.

\(^9\) [From Wikipedia] Poisson’s result was also derived in a previous study by Abraham de Moivre in 1711. Therefore some people suggest it should rightly be called the de Moivre distribution.
average flux of a region in the dataset with no targets. Its physical origin can be the brightness of the atmosphere (for ground-based instruments), possible stray light within the imaging instrument, the average flux of undetected targets, etc. The Sky value is thus an ideal definition, because in real datasets, what lies deep in the noise (far lower than the detection limit) is never known\(^{10}\). To account for all of these, the sky value is defined to be the average count/value of the undetected regions in the image. In a mock image/dataset, we have the luxury of setting the background (Sky) value.

In summary, the value in each element of the dataset (pixel in an image) is the sum of contributions from various galaxies and stars (after convolution by the PSF, see Section 8.1.1.2 [Point spread function], page 442). Let’s name the convolved sum of possibly overlapping objects in each pixel as \(I_{nn}\). \(nn\) represents ‘no noise’. For now, let’s assume the background \((B)\) is constant and sufficiently high for the Poisson distribution to be approximated by a Gaussian. Then the flux of that pixel, after adding noise, is a random value taken from a Gaussian distribution with the following mean \((\mu)\) and standard deviation \((\sigma)\):

\[
\mu = B + I_{nn}, \quad \sigma = \sqrt{B + I_{nn}}
\]

In astronomical instruments, \(B\) is enhanced by adding a “bias” level to each pixel before the shutter is even opened (for the exposure to start). As the exposure is ongoing and photo-electrons are accumulating from the astronomical objects, a “dark” current (due to thermal radiation of the instrument) also builds up in the pixels. The “dark” current will accumulate even when the shutter is closed, but the CCD electronics are working (hence the name “dark”). This added dark level further enhances the mean value in a real observation compared to the raw background value (from the atmosphere for example).

Since this type of noise is inherent in the objects we study, it is usually measured on the same scale as the astronomical objects, namely the magnitude system, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394. It is then internally converted to the flux scale for further processing.

The equations above clearly show the importance of the background value and its effect on the final signal to noise ratio in each pixel of a science image. It is therefore, one of the most important factors in understanding the noise (and properly simulating observations where necessary). An inappropriately bright background value can hide the signal of the mock profile hide behind the noise. In other words, a brighter background has larger standard deviation and vice versa. As a result, the only necessary parameter to define photon-counting noise over a mock image of simulated profiles is the background. For a complete example, see Section 2.1 [Sufi simulates a detection], page 18.

To better understand the correlation between the mean (or background) value and the noise standard deviation, let’s use an analogy. Consider the profile of your galaxy to be analogous to the profile of a ship that is sailing in the sea. The height of the ship would therefore be analogous to the maximum flux difference between your galaxy’s minimum and

\(^{10}\) In a real image, a relatively large number of very faint objects can be fully buried in the noise and never detected. These undetected objects will bias the background measurement to slightly larger values. Our best approximation is thus to simply assume they are uniform, and consider their average effect. See Figure 1 (a.1 and a.2) and Section 2.2 in Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664).
maximum values. Furthermore, let’s take the depth of the sea to represent the background value: a deeper sea, corresponds to a brighter background. In this analogy, the “noise” would be the height of the waves that surround the ship: in deeper waters, the waves would also be taller (the square root of the mean depth at the ship’s position).

If the ship is in deep waters, the height of waves are greater than when the ship is near to the beach (at lower depths). Therefore, when the ship is in the middle of the sea, there are high waves that are capable of hiding a significant part of the ship from our perspective. This corresponds to a brighter background value in astronomical images: the resulting noise from that brighter background can completely wash out the signal from a fainter galaxy, star or solar system object.

8.2.1.2 Instrumental noise

While taking images with a camera, a bias current is fed to the pixels, the variation of the value of this bias current over the pixels, also adds to the final image noise. Another source of noise is the readout noise that is produced by the electronics in the detector. Specifically, the parts that attempt to digitize the voltage produced by the photo-electrons in the analog to digital converter. With the current generation of instruments, this source of noise is not as significant as the noise due to the background Sky discussed in Section 8.2.1.1 [Photon counting noise], page 462.

Let $C$ represent the combined standard deviation of all these instrumental sources of noise. When only this source of noise is present, the noised pixel value would be a random value chosen from a Gaussian distribution with

$$
\mu = I_{nn}, \quad \sigma = \sqrt{C^2 + I_{nn}}
$$

This type of noise is independent of the signal in the dataset, it is only determined by the instrument. So the flux scale (and not magnitude scale) is most commonly used for this type of noise. In practice, this value is usually reported in analog-to-digital units or ADUs, not flux or electron counts. The gain value of the device can be used to convert between these two, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

8.2.1.3 Final noised pixel value

Based on the discussions in Section 8.2.1.1 [Photon counting noise], page 462, and Section 8.2.1.2 [Instrumental noise], page 464, depending on the values you specify for $B$ and $C$ from the above, the final noised value for each pixel is a random value chosen from a Gaussian distribution with

$$
\mu = B + I_{nn}, \quad \sigma = \sqrt{C^2 + B + I_{nn}}
$$

8.2.1.4 Generating random numbers

As discussed above, to generate noise we need to make random samples of a particular distribution. So it is important to understand some general concepts regarding the generation of random numbers. For a very complete and nice introduction we strongly advise reading
Donald Knuth’s “The art of computer programming”, volume 2, chapter 311. Quoting from the GNU Scientific Library manual, “If you don’t own it, you should stop reading right now, run to the nearest bookstore, and buy it”12!

Using only software, we can only produce what is called a pseudo-random sequence of numbers. A true random number generator is a hardware (let’s assume we have made sure it has no systematic biases), for example throwing dice or flipping coins (which have remained from the ancient times). More modern hardware methods use atmospheric noise, thermal noise or other types of external electromagnetic or quantum phenomena. All pseudo-random number generators (software) require a seed to be the basis of the generation. The advantage of having a seed is that if you specify the same seed for multiple runs, you will get an identical sequence of random numbers which allows you to reproduce the same final noised image.

The programs in GNU Astronomy Utilities (for example MakeNoise or MakeProfiles) use the GNU Scientific Library (GSL) to generate random numbers. GSL allows the user to set the random number generator through environment variables, see Section 3.3.1.2 [Installation directory], page 139, for an introduction to environment variables. In the chapter titled “Random Number Generation” they have fully explained the various random number generators that are available (there are a lot of them!). Through the two environment variables GSL_RNG_TYPE and GSL_RNG_SEED you can specify the generator and its seed respectively.

If you don’t specify a value for GSL_RNG_TYPE, GSL will use its default random number generator type. The default type is sufficient for most general applications. If no value is given for the GSL_RNG_SEED environment variable and you have asked Gnuastro to read the seed from the environment (through the --envseed option), then GSL will use the default value of each generator to give identical outputs. If you don’t explicitly tell Gnuastro programs to read the seed value from the environment variable, then they will use the system time (accurate to within a microsecond) to generate (apparently random) seeds. In this manner, every time you run the program, you will get a different random number distribution.

There are two ways you can specify values for these environment variables. You can call them on the same command-line for example:

```
$ GSL_RNG_TYPE="taus" GSL_RNG_SEED=345 astmknoise input.fits
```

In this manner the values will only be used for this particular execution of MakeNoise. Alternatively, you can define them for the full period of your terminal session or script length, using the shell’s export command with the two separate commands below (for a script remove the $ signs):

```
$ export GSL_RNG_TYPE="taus"
$ export GSL_RNG_SEED=345
```

The subsequent programs which use GSL’s random number generators will henceforth use these values in this session of the terminal you are running or while executing this script. In case you want to set fixed values for these parameters every time you use the GSL random number generator, you can add these two lines to your .bashrc startup script13, see Section 3.3.1.2 [Installation directory], page 139.

---

12 For students, running to the library might be more affordable!
13 Don’t forget that if you are going to give your scripts (that use the GSL random number generator) to others you have to make sure you also tell them to set these environment variable separately. So
IMPORTANT NOTE: If the two environment variables GSL_RNG_TYPE and GSL_RNG_SEED are defined, GSL will report them by default, even if you don’t use the --envseed option. For example, see this call to MakeProfiles:

```
$ export GSL_RNG_TYPE=taus
$ export GSL_RNG_SEED=345
$ astmkprof -s1 --kernel=gaussian,2,5
GSL_RNG_TYPE=taus
GSL_RNG_SEED=345
```

MakeProfiles V.VV started on DDD MMM DDD HH:MM:SS YYYY
- Building one gaussian kernel
- Random number generator (RNG) type: taus
- Basic RNG seed: 1618960836
---- ./kernel.fits created.
-- Output: ./kernel.fits
MakeProfiles finished in 0.068945 seconds

The first two output lines (showing the names and values of the GSL environment variables) are printed by GSL before MakeProfiles actually starts generating random numbers. Gnuastro’s programs will report the actual values they use independently (after the name of the program), you should check them for the final values used, not GSL’s printed values. In the example above, did you notice how the random number generator seed above is different between GSL and MakeProfiles? However, if --envseed was given, both printed seeds would be the same.

8.2.2 Invoking MakeNoise

MakeNoise will add noise to an existing image. The executable name is astmknoise with the following general template

```
$ astmknoise [OPTION ...] InputImage.fits
```

One line examples:

```
## Add noise with a standard deviation of 100 to image.
## (this is independent of the pixel value: not Poission noise)
$ astmknoise --sigma=100 image.fits

## Add noise to input image assuming a background magnitude (with
## zero point magnitude of 0) and a certain instrumental noise:
$ astmknoise --background=-10 -z0 --instrumental=20 mockimage.fits
```

If actual processing is to be done, the input image is a mandatory argument. The full list of options common to all the programs in Gnuastro can be seen in Section 4.1.2 [Common options], page 156. The type (see Section 4.5 [Numeric data types], page 178) of the output can be specified with the --type option, see Section 4.1.2.1 [Input/Output options], page 157. The header of the output FITS file keeps all the parameters that were influential in making it. This is done for future reproducibility.

---

for scripts, it is best to keep all such variable definitions within the script, even if they are within your .bashrc.
Chapter 8: Modeling and fitting

-b FLT
--background=FLT
The background value (per pixel) that will be added to each pixel value (internally) to simulate Poisson noise, see Section 8.2.1.1 [Photon counting noise], page 462. By default the units of this value are assumed to be in magnitudes, hence a --zeropoint is also necessary. If the background is in units of counts, you need add --bgisbrightness, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

Internally, the value given to this option will be converted to counts ($b$, when --bgisbrightness is called, the value will be used directly). Assuming the pixel value is $p$, the random value for that pixel will be taken from a Gaussian distribution with mean of $p + b$ and standard deviation of $\sqrt{p + b}$. With this option, the noise will therefore be dependent on the pixel values: according to the Poisson noise model, as the pixel value becomes larger, its noise will also become larger. This is thus a realistic way to model noise, see Section 8.2.1.1 [Photon counting noise], page 462.

-B
--bgisbrightness
The value given to --background should be interpreted as brightness, not as a magnitude.

-z FLT
--zeropoint=FLT
The zero point magnitude used to convert the value of --background (in units of magnitude) to flux, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

-i FLT
--instrumental=FLT
The instrumental noise which is in units of flux, see Section 8.2.1.2 [Instrumental noise], page 464.

-s FLT
--sigma=FLT
The total noise sigma in the same units as the pixel values. With this option, the --background, --zeropoint and --instrumental will be ignored. With this option, the noise will be independent of the pixel values (which is not realistic, see Section 8.2.1.1 [Photon counting noise], page 462). Hence it is only useful if you are working on low surface brightness regions where the change in pixel value (and thus real noise) is insignificant.

Generally, usage of this option is discouraged unless you understand the risks of not simulating real noise. This is because with this option, you will not get Poisson noise (the common noise model for astronomical imaging), where the noise varies based on pixel value. Use --background for adding Poission noise.
-e
--envseed
Use the GSL_RNG_SEED environment variable for the seed used in the random number generator, see Section 8.2.1.4 [Generating random numbers], page 464. With this option, the output image noise is always going to be identical (or reproducible).

-d
--doubletype
Save the output in the double precision floating point format that was used internally. This option will be most useful if the input images were of integer types.
9 High-level calculations

After the reduction of raw data (for example with the programs in Chapter 6 [Data manipulation], page 259) you will have reduced images/data ready for processing/analyzing (for example with the programs in Chapter 7 [Data analysis], page 337). But the processed/analyzed data (or catalogs) are still not enough to derive any scientific result. Even higher-level analysis is still needed to convert the observed magnitudes, sizes or volumes into physical quantities that we associate with each catalog entry or detected object which is the purpose of the tools in this section.

9.1 CosmicCalculator

To derive higher-level information regarding our sources in extra-galactic astronomy, cosmological calculations are necessary. In Gnuastro, CosmicCalculator is in charge of such calculations. Before discussing how CosmicCalculator is called and operates (in Section 9.1.3 [Invoking CosmicCalculator], page 474), it is important to provide a rough but mostly self-sufficient review of the basics and the equations used in the analysis. In Section 9.1.1 [Distance on a 2D curved space], page 469, the basic idea of understanding distances in a curved and expanding 2D universe (which we can visualize) are reviewed. Having solidified the concepts there, in Section 9.1.2 [Extending distance concepts to 3D], page 474, the formalism is extended to the 3D universe we are trying to study in our research.

The focus here is obtaining a physical insight into these equations (mainly for the use in real observational studies). There are many books thoroughly deriving and proving all the equations with all possible initial conditions and assumptions for any abstract universe, interested readers can study those books.

9.1.1 Distance on a 2D curved space

The observations to date (for example the Planck 2015 results), have not measured the presence of significant curvature in the universe. However to be generic (and allow its measurement if it does in fact exist), it is very important to create a framework that allows non-zero uniform curvature. However, this section is not intended to be a fully thorough and mathematically complete derivation of these concepts. There are many references available for such reviews that go deep into the abstract mathematical proofs. The emphasis here is on visualization of the concepts for a beginner.

As 3D beings, it is difficult for us to mentally create (visualize) a picture of the curvature of a 3D volume. Hence, here we will assume a 2D surface/space and discuss distances on that 2D surface when it is flat and when it is curved. Once the concepts have been created/visualized here, we will extend them, in Section 9.1.2 [Extending distance concepts to 3D], page 474, to a real 3D spatial slice of the Universe we live in and hope to study.

To be more understandable (actively discuss from an observer’s point of view) let’s assume there’s an imaginary 2D creature living on the 2D space (which might be curved in 3D). Here, we will be working with this creature in its efforts to analyze distances in its 2D universe. The start of the analysis might seem too mundane, but since it is difficult to

---

1 The observations are interpreted under the assumption of uniform curvature. For a relativistic alternative to dark energy (and maybe also some part of dark matter), non-uniform curvature may be even be more critical, but that is beyond the scope of this brief explanation.
imagine a 3D curved space, it is important to review all the very basic concepts thoroughly for an easy transition to a universe that is more difficult to visualize (a curved 3D space embedded in 4D).

To start, let’s assume a static (not expanding or shrinking), flat 2D surface similar to Figure 9.1 and that the 2D creature is observing its universe from point A. One of the most basic ways to parameterize this space is through the Cartesian coordinates \((x, y)\). In Figure 9.1, the basic axes of these two coordinates are plotted. An infinitesimal change in the direction of each axis is written as \(dx\) and \(dy\). For each point, the infinitesimal changes are parallel with the respective axes and are not shown for clarity. Another very useful way of parameterizing this space is through polar coordinates. For each point, we define a radius \((r)\) and angle \((\phi)\) from a fixed (but arbitrary) reference axis. In Figure 9.1 the infinitesimal changes for each polar coordinate are plotted for a random point and a dashed circle is shown for all points with the same radius.

![Figure 9.1: Two dimensional Cartesian and polar coordinates on a flat plane.](image)

Assuming an object is placed at a certain position, which can be parameterized as \((x, y)\), or \((r, \phi)\), a general infinitesimal change in its position will place it in the coordinates \((x + dx, y + dy)\), or \((r + dr, \phi + d\phi)\). The distance (on the flat 2D surface) that is covered by this infinitesimal change in the static universe \((ds_s)\), the subscript signifies the static nature of this universe) can be written as:

\[
ds_s^2 = dx^2 + dy^2 = dr^2 + r^2 d\phi^2
\]

The main question is this: how can the 2D creature incorporate the (possible) curvature in its universe when it’s calculating distances? The universe that it lives in might equally be a curved surface like Figure 9.2. The answer to this question but for a 3D being (us) is the whole purpose to this discussion. Here, we want to give the 2D creature (and later, ourselves) the tools to measure distances if the space (that hosts the objects) is curved.

Figure 9.2 assumes a spherical shell with radius \(R\) as the curved 2D plane for simplicity. The 2D plane is tangent to the spherical shell and only touches it at A. This idea will be generalized later. The first step in measuring the distance in a curved space is to imagine a third dimension along the \(z\) axis as shown in Figure 9.2. For simplicity, the \(z\) axis is assumed to pass through the center of the spherical shell. Our imaginary 2D creature cannot visualize the third dimension or a curved 2D surface within it, so the remainder of this discussion is purely abstract for it (similar to us having difficulty in visualizing a 3D curved space in 4D). But since we are 3D creatures, we have the advantage of visualizing the following steps. Fortunately the 2D creature is already familiar with our mathematical constructs, so it can follow our reasoning.
With the third axis added, a generic infinitesimal change over the full 3D space corresponds to the distance:

\[ ds^2 = dx^2 + dy^2 + dz^2 = dr^2 + r^2 d\phi^2 + dz^2. \]

Figure 9.2: 2D spherical shell (centered on \( O \)) and flat plane (light gray) tangent to it at point \( A \).

It is very important to recognize that this change of distance is for any point in the 3D space, not just those changes that occur on the 2D spherical shell of Figure 9.2. Recall that our 2D friend can only do measurements on the 2D surfaces, not the full 3D space. So we have to constrain this general change to any change on the 2D spherical shell. To do that, let’s look at the arbitrary point \( P \) on the 2D spherical shell. Its image (\( P' \)) on the flat plain is also displayed. From the dark gray triangle, we see that

\[
\sin \theta = \frac{r}{R}, \quad \cos \theta = \frac{R - z}{R}.
\]

These relations allow the 2D creature to find the value of \( z \) (an abstract dimension for it) as a function of \( r \) (distance on a flat 2D plane, which it can visualize) and thus eliminate \( z \). From \( \sin^2 \theta + \cos^2 \theta = 1 \), we get \( z^2 - 2Rz + r^2 = 0 \) and solving for \( z \), we find:

\[
z = R \left( 1 \pm \sqrt{1 - \frac{r^2}{R^2}} \right).
\]
The ± can be understood from Figure 9.2: For each \( r \), there are two points on the sphere, one in the upper hemisphere and one in the lower hemisphere. An infinitesimal change in \( r \), will create the following infinitesimal change in \( z \):

\[
\mathrm{d}z = \pm \frac{r}{R} \left( \frac{1}{\sqrt{1 - r^2/R^2}} \right) \mathrm{d}r.
\]

Using the positive signed equation instead of \( \mathrm{d}z \) in the \( ds^2 \) equation above, we get:

\[
ds^2_s = \frac{\mathrm{d}r^2}{1 - r^2/R^2} + r^2 \mathrm{d}\phi^2.
\]

The derivation above was done for a spherical shell of radius \( R \) as a curved 2D surface. To generalize it to any surface, we can define \( K = 1/R^2 \) as the curvature parameter. Then the general infinitesimal change in a static universe can be written as:

\[
ds_s^2 = \frac{\mathrm{d}r^2}{1 - Kr^2} + r^2 \mathrm{d}\phi^2.
\]

Therefore, when \( K > 0 \) (and curvature is the same everywhere), we have a finite universe, where \( r \) cannot become larger than \( R \) as in Figure 9.2. When \( K = 0 \), we have a flat plane (Figure 9.1) and a negative \( K \) will correspond to an imaginary \( R \). The latter two cases may be infinite in area (which is not a simple concept, but mathematically can be modeled with \( r \) extending infinitely), or finite-area (like a cylinder is flat everywhere with \( ds_s^2 = \mathrm{d}x^2 + \mathrm{d}y^2 \), but finite in one direction in size).

A very important issue that can be discussed now (while we are still in 2D and can actually visualize things) is that \( \overrightarrow{P} \) is tangent to the curved space at the observer’s position. In other words, it is on the gray flat surface of Figure 9.2, even when the universe if curved: \( \overrightarrow{P} = P' - A \). Therefore for the point \( P \) on a curved space, the raw coordinate \( r \) is the distance to \( P' \), not \( P \). The distance to the point \( P \) (at a specific coordinate \( r \) on the flat plane) over the curved surface (thick line in Figure 9.2) is called the proper distance and is displayed with \( l \). For the specific example of Figure 9.2, the proper distance can be calculated with: \( l = R\theta \) (\( \theta \) is in radians). Using the \( \sin \theta \) relation found above, we can find \( l \) as a function of \( r \):

\[
\theta = \sin^{-1} \left( \frac{r}{R} \right) \rightarrow l(r) = R\sin^{-1} \left( \frac{r}{R} \right)
\]

\( R \) is just an arbitrary constant and can be directly found from \( K \), so for cleaner equations, it is common practice to set \( R = 1 \), which gives: \( l(r) = \sin^{-1} r \). Also note that when \( R = 1 \), then \( l = \theta \). Generally, depending on the curvature, in a static universe the proper distance can be written as a function of the coordinate \( r \) as (from now on we are assuming \( R = 1 \)):

\[
l(r) = \sin^{-1}(r) \quad (K > 0), \quad l(r) = r \quad (K = 0), \quad l(r) = \sinh^{-1}(r) \quad (K < 0).
\]
With \( l \), the infinitesimal change of distance can be written in a more simpler and abstract form of
\[
\frac{ds_{s}^{2}}{s} = dl^{2} + r^{2}d\phi^{2}.
\]

Until now, we had assumed a static universe (not changing with time). But our observations so far appear to indicate that the universe is expanding (it isn’t static). Since there is no reason to expect the observed expansion is unique to our particular position of the universe, we expect the universe to be expanding at all points with the same rate at the same time. Therefore, to add a time dependence to our distance measurements, we can include a multiplicative scaling factor, which is a function of time: \( a(t) \). The functional form of \( a(t) \) comes from the cosmology, the physics we assume for it: general relativity, and the choice of whether the universe is uniform (‘homogeneous’) in density and curvature or inhomogeneous. In this section, the functional form of \( a(t) \) is irrelevant, so we can avoid these issues.

With this scaling factor, the proper distance will also depend on time. As the universe expands, the distance between two given points will shift to larger values. We thus define a distance measure, or coordinate, that is independent of time and thus doesn’t ‘move’. We call it the **comoving distance** and display with \( \chi \) such that: \( l(r,t) = \chi(r)a(t) \). We have therefore, shifted the \( r \) dependence of the proper distance we derived above for a static universe to the comoving distance:

\[
\chi(r) = \sin^{-1}(r) \quad (K > 0), \quad \chi(r) = r \quad (K = 0), \quad \chi(r) = \sinh^{-1}(r) \quad (K < 0).
\]

Therefore, \( \chi(r) \) is the proper distance to an object at a specific reference time: \( t = t_{r} \) (the \( r \) subscript signifies “reference”) when \( a(t_{r}) = 1 \). At any arbitrary moment \((t \neq t_{r})\) before or after \( t_{r} \), the proper distance to the object can be scaled with \( a(t) \).

Measuring the change of distance in a time-dependent (expanding) universe only makes sense if we can add up space and time\(^2\). But we can only add bits of space and time together if we measure them in the same units: with a conversion constant (similar to how 1000 is used to convert a kilometer into meters). Experimentally, we find strong support for the hypothesis that this conversion constant is the speed of light (or gravitational waves\(^3\)) in a vacuum. This speed is postulated to be constant\(^4\) and is almost always written as \( c \). We can thus parameterize the change in distance on an expanding 2D surface as
\[
\frac{ds^{2}}{s} = c^{2}dt^{2} - a^{2}(t)\frac{ds_{s}^{2}}{s} = c^{2}dt^{2} - a^{2}(t)(d\chi^{2} + r^{2}d\phi^{2}).
\]

\(^{2}\) In other words, making our space-time consistent with Minkowski space-time geometry. In this geometry, different observers at a given point (event) in space-time split up space-time into ‘space’ and ‘time’ in different ways, just like people at the same spatial position can make different choices of splitting up a map into ‘left–right’ and ‘up–down’. This model is well supported by twentieth and twenty-first century observations.

\(^{3}\) The speed of gravitational waves was recently found to be very similar to that of light in vacuum, see arXiv:1710.05834 (https://arxiv.org/abs/1710.05834).

\(^{4}\) In natural units, speed is measured in units of the speed of light in vacuum.
9.1.2 Extending distance concepts to 3D

The concepts of Section 9.1.1 [Distance on a 2D curved space], page 469, are here extended to a 3D space that might be curved. We can start with the generic infinitesimal distance in a static 3D universe, but this time in spherical coordinates instead of polar coordinates. \( \theta \) is shown in Figure 9.2, but here we are 3D beings, positioned on \( O \) (the center of the sphere) and the point \( O \) is tangent to a 4D-sphere. In our 3D space, a generic infinitesimal displacement will correspond to the following distance in spherical coordinates:

\[
    ds^2 = dr^2 + r^2 (d\theta^2 + \sin^2 \theta d\phi^2).
\]

Like the 2D creature before, we now have to assume an abstract dimension which we cannot visualize easily. Let’s call the fourth dimension \( w \), then the general change in coordinates in the full four dimensional space will be:

\[
    ds^2 = dr^2 + r^2 (d\theta^2 + \sin^2 \theta d\phi^2) + dw^2.
\]

But we can only work on a 3D curved space, so following exactly the same steps and conventions as our 2D friend, we arrive at:

\[
    ds^2 = \frac{dr^2}{1 - Kr^2} + r^2 (d\theta^2 + \sin^2 \theta d\phi^2).
\]

In a non-static universe (with a scale factor \( a(t) \)), the distance can be written as:

\[
    ds^2 = c^2 dt^2 - a^2(t) [d\chi^2 + r^2 (d\theta^2 + \sin^2 \theta d\phi^2)].
\]

9.1.3 Invoking CosmicCalculator

CosmicCalculator will calculate cosmological variables based on the input parameters. The executable name is `astcosmiccal` with the following general template

\$ astcosmiccal [OPTION...] ...

One line examples:

## Print basic cosmological properties at redshift 2.5:
\$ astcosmiccal -z2.5

## Only print Comoving volume over 4pi stradian to z (Mpc\(^3\)):
\$ astcosmiccal --redshift=0.8 --volume

## Print redshift and age of universe when Lyman-alpha line is
## at 6000 angstrom (another way to specify redshift).
\$ astcosmiccal --obsline=lyalpha,6000 --age

## Print luminosity distance, angular diameter distance and age
## of universe in one row at redshift 0.4

$\text{astcosmiccal -z0.4 -LA}g$

## Assume Lambda and matter density of 0.7 and 0.3 and print
## basic cosmological parameters for redshift 2.1:

$\text{astcosmiccal -l0.7 -m0.3 -z2.1}$

## Print wavelength of all pre-defined spectral lines when
## Lyman-alpha is observed at 4000 Angstroms.

$\text{astcosmiccal --obsline=lyalpha,4000 --listlinesatz}$

The input parameters (for example current matter density, etc) can be given as
command-line options or in the configuration files, see Section 4.2 [Configuration files],
page 169. For a definition of the different parameters, please see the sections prior to this.
If no redshift is given, CosmicCalculator will just print its input parameters and abort.
For a full list of the input options, please see Section 9.1.3.1 [CosmicCalculator input
options], page 475.

Without any particular output requested (and only a given redshift), CosmicCalculator
will print all basic cosmological calculations (one per line) with some explanations before
each. This can be good when you want a general feeling of the conditions at a specific
redshift. Alternatively, if any specific calculation(s) are requested (its possible to call more
than one), only the requested value(s) will be calculated and printed with one character
space between them. In this case, no description or units will be printed. See Section 9.1.3.2
[CosmicCalculator basic cosmology calculations], page 478, for the full list of these options
along with some explanations how when/how they can be useful.

Another common operation in observational cosmology is dealing with spectral lines at
different redshifts. CosmicCalculator also has features to help in such situations, please see
Section 9.1.3.3 [CosmicCalculator spectral line calculations], page 480.

### 9.1.3.1 CosmicCalculator input options

The inputs to CosmicCalculator can be specified with the following options:

-`-z FLT`
-`--redshift=FLT`

The redshift of interest. There are two other ways that you can specify the
target redshift: 1) Spectral lines and their observed wavelengths, see `--obsline`.
2) Velocity, see `--velocity`. Hence this option cannot be called with `--obsline`
or `--velocity`.

-`-y FLT`
-`--velocity=FLT`

Input velocity in km/s. The given value will be converted to redshift internally,
and used in any subsequent calculation. This option is thus an alternative to
`--redshift` or `--obsline`, it cannot be used with them. The conversion will
be done with the more general and accurate relativistic equation of $1 + z = \sqrt{(c + v)/(c - v)}$, not the simplified $z \approx v/c$.

-`-H FLT`
-`--H0=FLT`

Current expansion rate (in km sec$^{-1}$ Mpc$^{-1}$).
-l FLT
--olambda=FLT
Cosmological constant density divided by the critical density in the current Universe ($\Omega_{\Lambda,0}$).

-m FLT
--omatter=FLT
Matter (including massive neutrinos) density divided by the critical density in the current Universe ($\Omega_{m,0}$).

-r FLT
--oradiation=FLT
Radiation density divided by the critical density in the current Universe ($\Omega_{r,0}$).

-0 STR/FLT,FLT
--obsline=STR/FLT,FLT
Find the redshift to use in next steps based on the rest-frame and observed wavelengths of a line. This option is thus an alternative to --redshift or --velocity, it cannot be used with them. Wavelengths are assumed to be in Angstroms. The first argument identifies the line. It can be one of the standard names below, or any rest-frame wavelength in Angstroms. The second argument is the observed wavelength of that line. For example --obsline=lyalpha,6000 is the same as --obsline=1215.64,6000.

The pre-defined names are listed below, sorted from red (longer wavelength) to blue (shorter wavelength). You can get this list on the command-line with the --listlines.

- siired $[6731 \text{Å}]$ SII doublet’s redder line.
- sii $[6724 \text{Å}]$ SII doublet’s mean center at .
- siiblue $[6717 \text{Å}]$ SII doublet’s bluer line.
- niired $[6584 \text{Å}]$ NII doublet’s redder line.
- nii $[6566 \text{Å}]$ NII doublet’s mean center.
- halpha $[6562.8 \text{Å}]$ H-α line.
- niiblue $[6548 \text{Å}]$ NII doublet’s bluer line.
- oiiired-vis $[5007 \text{Å}]$ OIII doublet’s redder line in the visible.
- oiii-vis $[4983 \text{Å}]$ OIII doublet’s mean center in the visible.
- oiiiblue-vis $[4959 \text{Å}]$ OIII doublet’s bluer line in the visible.
- hbeta $[4861.36 \text{Å}]$ H-β line.
- heii-vis $[4686 \text{Å}]$ HeII doublet’s redder line in the visible.
- hgamma $[4340.46 \text{Å}]$ H-γ line.
- hdelta $[4101.74 \text{Å}]$ H-δ line.
hepsilon $[3970.07\,\text{Å}]$ H-\(\epsilon\) line.
neiii $[3869\,\text{Å}]$ NEIII line.
oiired $[3729\,\text{Å}]$ OII doublet’s redder line.
oii $[3727.5\,\text{Å}]$ OII doublet’s mean center.
oiiblue $[3726\,\text{Å}]$ OII doublet’s bluer line.
blimit $[3646\,\text{Å}]$ Balmer limit.
mgiired $[2803\,\text{Å}]$ MgII doublet’s redder line.
mgi $[2799.5\,\text{Å}]$ MgII doublet’s mean center.
mgiiblue $[2796\,\text{Å}]$ MgII doublet’s bluer line.
ciiired $[1909\,\text{Å}]$ CIII doublet’s redder line.
ciii $[1908\,\text{Å}]$ CIII doublet’s mean center.
ciiiblue $[1907\,\text{Å}]$ CIII doublet’s bluer line.
si_iiired $[1892\,\text{Å}]$ SiIII doublet’s redder line.
si_iii $[1887.5\,\text{Å}]$ SiIII doublet’s mean center.
si_iiblue $[1883\,\text{Å}]$ SiIII doublet’s bluer line.
oiiired-uv $[1666\,\text{Å}]$ OIII doublet’s redder line in the ultra-violet.
oiii-uv $[1663.5\,\text{Å}]$ OIII doublet’s mean center in the ultra-violet.
oiiiblue-uv $[1661\,\text{Å}]$ OIII doublet’s bluer line in the ultra-violet.
heii-uv $[1640\,\text{Å}]$ HeII doublet’s bluer line in the ultra-violet.
civred $[1551\,\text{Å}]$ CIV doublet’s redder line.
civ $[1549\,\text{Å}]$ CIV doublet’s mean center.
civblue $[1548\,\text{Å}]$ CIV doublet’s bluer line.
nv $[1240\,\text{Å}]$ NV (four times ionized Sodium).
lyalpha $[1215.67\,\text{Å}]$ Lyman-\(\alpha\) line.
lybeta $[1025.7\,\text{Å}]$ Lyman-\(\beta\) line.
lygamma $[972.54\,\text{Å}]$ Lyman-\(\gamma\) line.
lydelta $[949.74\,\text{Å}]$ Lyman-\(\delta\) line.
lyepsilon $[937.80\,\text{Å}]$ Lyman-\(\epsilon\) line.
lylimit $[912\,\text{Å}]$ Lyman limit.
9.1.3.2 CosmicCalculator basic cosmology calculations

By default, when no specific calculations are requested, CosmicCalculator will print a complete set of all its calculators (one line for each calculation, see Section 9.1.3 [Invoking CosmicCalculator], page 474). The full list of calculations can be useful when you don’t want any specific value, but just a general view. In other contexts (for example in a batch script or during a discussion), you know exactly what you want and don’t want to be distracted by all the extra information.

You can use any number of the options described below in any order. When any of these options are requested, CosmicCalculator’s output will just be a single line with a single space between the (possibly) multiple values. In the example below, only the tangential distance along one arc-second (in kpc), absolute magnitude conversion, and age of the universe at redshift 2 are printed (recall that you can merge short options together, see Section 4.1.1.2 [Options], page 154).

```
$ astcosmiccal -z2 -sag
8.585046 44.819248 3.289979
```

Here is one example of using this feature in scripts: by adding the following two lines in a script to keep/use the comoving volume with varying redshifts:

```c
z=3.12
vol=$(astcosmiccal --redshift=$z --volume)
```

In a script, this operation might be necessary for a large number of objects (several of galaxies in a catalog for example). So the fact that all the other default calculations are ignored will also help you get to your result faster.

If you are indeed dealing with many (for example thousands) of redshifts, using CosmicCalculator is not the best/fastest solution. Because it has to go through all the configuration files and preparations for each invocation. To get the best efficiency (least overhead), we recommend using Gnuastro’s cosmology library (see Section 11.3.30 [Cosmology library (cosmology.h)], page 660). CosmicCalculator also calls the library functions defined there for its calculations, so you get the same result with no overhead. Gnuastro also has libraries for easily reading tables into a C program, see Section 11.3.10 [Table input output (table.h)], page 568. Afterwards, you can easily build and run your C program for the particular processing with Section 11.2 [BuildProgram], page 521.

If you just want to inspect the value of a variable visually, the description (which comes with units) might be more useful. In such cases, the following command might be better. The other calculations will also be done, but they are so fast that you will not notice on modern computers (the time it takes your eye to focus on the result is usually longer than the processing: a fraction of a second).

```
$ astcosmiccal --redshift=0.832 | grep volume
```

The full list of CosmicCalculator’s specific calculations is present below in two groups: basic cosmology calculations and those related to spectral lines. In case you have forgot the units, you can use the --help option which has the units along with a short description.

-e
--usedredshift

The redshift that was used in this run. In many cases this is the main input parameter to CosmicCalculator, but it is useful in others. For example
in combination with --obsline (where you give an observed and rest-frame wavelength and would like to know the redshift) or with --velocity (where you specify the velocity instead of redshift). Another example is when you run CosmicCalculator in a loop, while changing the redshift and you want to keep the redshift value with the resulting calculation.

-Y
--usedvelocity
The velocity (in km/s) that was used in this run. The conversion from redshift will be done with the more general and accurate relativistic equation of $1 + z = \sqrt{(c + v)/(c - v)}$, not the simplified $z \approx v/c$.

-G
--agenow
The current age of the universe (given the input parameters) in Ga (Giga annum, or billion years).

-C
--criticaldensitynow
The current critical density (given the input parameters) in grams per centimeter-cube ($g/cm^3$).

-d
--properdistance
The proper distance (at current time) to object at the given redshift in Megaparsecs (Mpc). See Section 9.1.1 [Distance on a 2D curved space], page 469, for a description of the proper distance.

-A
--angulardimdist
The angular diameter distance to object at given redshift in Megaparsecs (Mpc).

-s
--arcsectandist
The tangential distance covered by 1 arc-seconds at the given redshift in kiloparsecs (Kpc). This can be useful when trying to estimate the resolution or pixel scale of an instrument (usually in units of arc-seconds) at a given redshift.

-L
--luminositydist
The luminosity distance to object at given redshift in Megaparsecs (Mpc).

-u
--distancemodulus
The distance modulus at given redshift.

-a
--absmagconv
The conversion factor (addition) to absolute magnitude. Note that this is practically the distance modulus added with $-2.5 \log (1 + z)$ for the desired redshift based on the input parameters. Once the apparent magnitude and redshift of an object is known, this value may be added with the apparent magnitude to give the object’s absolute magnitude.
--age  
Age of the universe at given redshift in Ga (Giga annum, or billion years).

--lookbacktime  
The look-back time to given redshift in Ga (Giga annum, or billion years). The look-back time at a given redshift is defined as the current age of the universe (--agenow) subtracted by the age of the universe at the given redshift.

--criticaldensity  
The critical density at given redshift in grams per centimeter-cube ($g/cm^3$).

--onlyvolume  
The comoving volume in Megaparsecs cube ($Mpc^3$) until the desired redshift based on the input parameters.

9.1.3.3 CosmicCalculator spectral line calculations
At different redshifts, observed spectral lines are shifted compared to their rest frame wavelengths with this simple relation: $\lambda_{\text{obs}} = \lambda_{\text{rest}}(1 + z)$. Although this relation is very simple and can be done for one line in the head (or a simple calculator!), it slowly becomes tiring when dealing with a lot of lines or redshifts, or some precision is necessary. The options in this section are thus provided to greatly simplify usage of this simple equation, and also helping by storing a list of pre-defined spectral line wavelengths.

For example if you want to know the wavelength of the $H\alpha$ line (at 6562.8 Angstroms in rest frame), when $Ly\alpha$ is at 8000 Angstroms, you can call CosmicCalculator like the first example below. And if you want the wavelength of all pre-defined spectral lines at this redshift, you can use the second command.

```bash
$ astcosmiccal --obsline=lyalpha,8000 --lineatz=halpha
$ astcosmiccal --obsline=lyalpha,8000 --listlinesatz
```

Below you can see the printed/output calculations of CosmicCalculator that are related to spectral lines. Note that --obsline is an input parameter, so its discussed (with the full list of known lines) in Section 9.1.3.1 [CosmicCalculator input options], page 475.

--listlines  
List the pre-defined rest frame spectral line wavelengths and their names on standard output, then abort CosmicCalculator. When this option is given, other operations on the command-line will be ignored. This is convenient when you forget the specific name of the spectral line used within Gnuastro, or when you forget the exact wavelength of a certain line.

These names can be used with the options that deal with spectral lines, for example --obsline and --lineatz (Section 9.1.3.2 [CosmicCalculator basic cosmology calculations], page 478).

The format of the output list is a two-column table, with Gnuastro’s text table format (see Section 4.7.2 [Gnuastro text table format], page 185). Therefore, if you are only looking for lines in a specific range, you can pipe the output into Gnuastro’s table program and use its --range option on the wavelength (first)
column. For example, if you only want to see the lines between 4000 and 6000 Angstroms, you can run this command:

```bash
$ astcosmiccal --listlines \
  | asttable --range=wavelength,4000,6000
```

And if you want to use the list later and have it as a table in a file, you can easily add the `--output` (or `-o`) option to the `asttable` command, and specify the filename, for example `--output=lines.fits` or `--output=lines.txt`.

`--listlinesatz`

Similar to `--listlines` (above), but the printed wavelength is not in the rest frame, but redshifted to the given redshift. Recall that the redshift can be specified by `--redshift` directly or by `--obsline`, see Section 9.1.3.1 [CosmicCalculator input options], page 475.

`-i STR/FLT`

`--lineatz=STR/FLT`

The wavelength of the specified line at the redshift given to CosmicCalculator. The line can be specified either by its name or directly as a number (its wavelength). To get the list of pre-defined names for the lines and their wavelength, you can use the `--listlines` option, see Section 9.1.3.1 [CosmicCalculator input options], page 475. In the former case (when a name is given), the returned number is in units of Angstroms. In the latter (when a number is given), the returned value is the same units of the input number (assuming its a wavelength).
10 Installed scripts

Gnuastro’s programs (introduced in previous chapters) are designed to be highly modular and thus contain lower-level operations on the data. However, in many contexts, certain higher-level are also shared between many contexts. For example a sequence of calls to multiple Gnuastro programs, or a special way of running a program and treating the output. To facilitate such higher-level data analysis, Gnuastro also installs some scripts on your system with the (astscript-) prefix (in contrast to the other programs that only have the ast prefix).

Like all of Gnuastro’s source code, these scripts are also heavily commented. They are written in portable shell scripts (command-line environments), which doesn’t need compilation. Therefore, if you open the installed scripts in a text editor, you can actually read them\(^1\). For example with this command (just replace nano with your favorite text editor, like emacs or vim):

\[\text{\$ nano $(which astscript-NAME)}\]

Shell scripting is the same language that you use when typing on the command-line. Therefore shell scripting is much more widely known and used compared to C (the language of other Gnuastro programs). Because Gnuastro’s installed scripts do higher-level operations, customizing these scripts for a special project will be more common than the programs.

These scripts also accept options and are in many ways similar to the programs (see Section 4.1.2 [Common options], page 156) with some minor differences:

- Currently they don’t accept configuration files themselves. However, the configuration files of the Gnuastro programs they call are indeed parsed and used by those programs. As a result, they don’t have the following options: `--checkconfig, --config, --lastconfig, --onlyversion, --printparams, --setdirconf and --setusrconf.`
- They don’t directly allocate any memory, so there is no `--minmapsize`.
- They don’t have an independent `--usage` option: when called with `--usage`, they just recommend running `--help`.
- The output of `--help` is not configurable like the programs (see Section 4.3.2 [--help], page 173).
- The scripts will commonly use your installed shell and other basic command-line tools (for example AWK or SED). Different systems have different versions and implementations of these basic tools (for example GNU/Linux systems use GNU Bash, GNU AWK and GNU SED which are far more advanced and up to date then the minimalist AWK and SED of most other systems). Therefore, unexpected errors in these tools might come up when you run these scripts on non-GNU/Linux operating systems. If you do confront such strange errors, please submit a bug report so we fix it as soon as possible (see Section 1.8 [Report a bug], page 11).

---

\(^1\) Gnuastro’s installed programs (those only starting with ast) aren’t human-readable. They are written in C and need to be compiled before execution. Compilation optimizes the steps into the low-level hardware CPU instructions/language to improve efficiency. Because compiled programs don’t need an interpreter like Bash on every run, they are much faster and more independent than scripts. To read the source code of the programs, look into the bin/progname directory of Gnuastro’s source (Section 3.2 [Downloading the source], page 131). If you would like to read more about why C was chosen for the programs, please see Section 12.1 [Why C programming language?], page 670.
10.1 Sort FITS files by night

FITS images usually contain (several) keywords for preserving important dates. In particular, for lower-level data, this is usually the observation date and time (for example, stored in the `DATE-OBS` keyword value). When analyzing observed datasets, many calibration steps (like the dark, bias or flat-field), are commonly calculated on a per-observing-night basis.

However, the FITS standard’s date format (`YYYY-MM-DDThh:mm:ss.ddd`) is based on the western (Gregorian) calendar. Dates that are stored in this format are complicated for automatic processing: a night starts in the final hours of one calendar day, and extends to the early hours of the next calendar day. As a result, to identify datasets from one night, we commonly need to search for two dates. However calendar peculiarities can make this identification very difficult. For example when an observation is done on the night separating two months (like the night starting on March 31st and going into April 1st), or two years (like the night starting on December 31st 2018 and going into January 1st, 2019). To account for such situations, it is necessary to keep track of how many days are in a month, and leap years, etc.

Gnuastro’s `astscript-sort-by-night` script is created to help in such important scenarios. It uses Section 5.1 [Fits], page 194, to convert the FITS date format into the Unix epoch time (number of seconds since 00:00:00 of January 1st, 1970), using the `--datetosec` option. The Unix epoch time is a single number (integer, if not given in sub-second precision), enabling easy comparison and sorting of dates after January 1st, 1970.

You can use this script as a basis for making a much more highly customized sorting script. Here are some examples

- If you need to copy the files, but only need a single extension (not the whole file), you can add a step just before the making of the symbolic links, or copies, and change it to only copy a certain extension of the FITS file using the Fits program’s `--copy` option, see Section 5.1.1.1 [HDU information and manipulation], page 197.

- If you need to classify the files with finer detail (for example the purpose of the dataset), you can add a step just before the making of the symbolic links, or copies, to specify a file-name prefix based on other certain keyword values in the files. For example when the FITS files have a keyword to specify if the dataset is a science, bias, or flat-field image. You can read it and to add a `sci-`, `bias-`, or `flat-` to the created file (after the `--prefix`) automatically.

For example, let’s assume the observing mode is stored in the hypothetical `MODE` keyword, which can have three values of `BIAS-IMAGE`, `SCIENCE-IMAGE` and `FLAT-EXP`. With the step below, you can generate a mode-prefix, and add it to the generated link/copy names (just correct the filename and extension of the first line to the script’s variables):

```bash
modepref=$(astfits infile.fits -h1 \
| sed -e "s/'/ /g" \n| awk "$1=="MODE{" \n  if($3=="BIAS-IMAGE") print "bias-"; \n  else if($3=="SCIENCE-IMAGE") print "sci-"; \n  else if($3=="FLAT-EXP") print "flat-"; \n  else print $3, "NOT recognized"; exit 1}"
```

Here is a description of it. We first use `astfits` to print all the keywords in extension 1 of `infile.fits`. In the FITS standard, string values (that we are assuming here) are
placed in single quotes ('') which are annoying in this context/use-case. Therefore, we pipe the output of astfits into sed to remove all such quotes (substituting them with a blank space). The result is then piped to AWK for giving us the final mode-prefix: with $1=="MODE"$, we ask AWK to only consider the line where the first column is MODE. There is an equal sign between the key name and value, so the value is the third column ($3$ in AWK). We thus use a simple if-else structure to look into this value and print our custom prefix based on it. The output of AWK is then stored in the modepref shell variable which you can add to the link/copy name.

With the solution above, the increment of the file counter for each night will be independent of the mode. If you want the counter to be mode-dependent, you can add a different counter for each mode and use that counter instead of the generic counter for each night (based on the value of modepref). But we’ll leave the implementation of this step to you as an exercise.

### 10.1.1 Invoking astscript-sort-by-night

This installed script will read a FITS date formatted value from the given keyword, and classify the input FITS files into individual nights. For more on installed scripts please see (see Chapter 10 [Installed scripts], page 482). This script can be used with the following general template:

```
$ astscript-sort-by-night [OPTION...] FITS-files
```

One line examples:

```bash
## Use the DATE-OBS keyword
$ astscript-sort-by-night --key=DATE-OBS /path/to/data/*.fits

## Make links to the input files with the 'img-' prefix
$ astscript-sort-by-night --link --prefix=img- /path/to/data/*.fits
```

This script will look into a HDU/extension (--hdu) for a keyword (--key) in the given FITS files and interpret the value as a date. The inputs will be separated by "night"s (11:00a.m to next day’s 10:59:59a.m, spanning two calendar days, exact hour can be set with --hour).

The default output is a list of all the input files along with the following two columns: night number and file number in that night (sorted by time). With --link a symbolic link will be made (one for each input) that contains the night number, and number of file in that night (sorted by time), see the description of --link for more. When --copy is used instead of a link, a copy of the inputs will be made instead of symbolic link.

Below you can see one example where all the target-*.fits files in the data directory should be separated by observing night according to the DATE-OBS keyword value in their second extension (number 1, recall that HDU counting starts from 0). You can see the output after the ls command.

```
$ astscript-sort-by-night -pimg- -h1 -kDATE-OBS data/target-*.fits
$ ls
img-n1-1.fits img-n1-2.fits img-n2-1.fits ...
```

The outputs can be placed in a different (already existing) directory by including that directory’s name in the --prefix value, for example --prefix=sorted/img- will put them all under the sorted directory.
This script can be configured like all Gnuastro’s programs (through command-line options, see Section 4.1.2 [Common options], page 156), with some minor differences that are described in Chapter 10 [Installed scripts], page 482. The particular options to this script are listed below:

- **h** STR
  --hdu=STR
  The HDU/extension to use in all the given FITS files. All of the given FITS files must have this extension.

- **k** STR
  --key=STR
  The keyword name that contains the FITS date format to classify/sort by.

- **H** FLT
  --hour=FLT
  The hour that defines the next “night”. By default, all times before 11:00a.m are considered to belong to the previous calendar night. If a sub-hour value is necessary, it should be given in units of hours, for example --hour=9.5 corresponds to 9:30a.m.

---

**Dealing with time zones:** The time that is recorded in --key may be in UTC (Universal Time Coordinate). However, the organization of the images taken during the night depends on the local time. It is possible to take this into account by setting the --hour option to the local time in UTC.

For example, consider a set of images taken in Auckland (New Zealand, UTC+12) during different nights. If you want to classify these images by night, you have to know at which time (in UTC time) the Sun rises (or any other separator/definition of a different night). For example if your observing night finishes before 9:00a.m in Auckland, you can use --hour=21. Because in Auckland the local time of 9:00 corresponds to 21:00 UTC.

---

- **l**
  --link
  Create a symbolic link for each input FITS file. This option cannot be used with --copy. The link will have a standard name in the following format (variable parts are written in CAPITAL letters and described after it):

  PnN-I.fits

  **P**
  This is the value given to --prefix. By default, its value is ./ (to store the links in the directory this script was run in). See the description of --prefix for more.

  **N**
  This is the night-counter: starting from 1. N is just incremented by 1 for the next night, no matter how many nights (without any dataset) there are between two subsequent observing nights (its just an identifier for each night which you can easily map to different calendar nights).

  **I**
  File counter in that night, sorted by time.
-c
--copy Make a copy of each input FITS file with the standard naming convention described in --link. With this option, instead of making a link, a copy is made. This option cannot be used with --link.

-p STR
--prefix=STR
Prefix to append before the night-identifier of each newly created link or copy. This option is thus only relevant with the --copy or --link options. See the description of --link for how its used. For example, with --prefix=img-, all the created file names in the current directory will start with img-, making outputs like img-n1-1.fits or img-n3-42.fits.
--prefix can also be used to store the links/copies in another directory relative to the directory this script is being run (it must already exist). For example --prefix=/path/to/processing/img- will put all the links/copies in the /path/to/processing directory, and the files (in that directory) will all start with img-.

--stdintimeout=INT
Number of micro-secs to wait for standard input within this script. This does not correspond to general inputs into the script, inputs to the script should always be given as a file. However, within the script, pipes are often used to pass the output of one program to another. The value given to this option will be passed to those internal pipes. When running this script, if you confront an error, saying “No input!”, you should be able to fix it by giving a larger number to this option (the default value is 1000000 micro-secs or 10 seconds).

10.2 Generate radial profile

The 1 dimensional radial profile of an object is an important parameter in many aspects of astronomical image processing. For example, you want to study how the light of a galaxy is distributed as a function of the radial distance from the center. In other cases, the radial profile of a star can show the PSF (see Section 8.1.1.2 [Point spread function], page 442). Gnuastro’s astscript-radial-profile script is created to obtain such radial profiles for one object within an image. This script uses Section 8.1 [MakeProfiles], page 440, to generate elliptical apertures with the values equal to the distance from the center of the object and Section 7.4 [MakeCatalog], page 392, for measuring the values over the apertures.

10.2.1 Invoking astscript-radial-profile

This installed script will measure the radial profile of an object within an image. For more on installed scripts please see (see Chapter 10 [Installed scripts], page 482). This script can be used with the following general template:

$ astscript-radial-profile [OPTION...] FITS-file

Examples:

## Generate the radial profile with default options (assuming the ## object is in the center of the image, and using the mean).
$ astscript-radial-profile image.fits
## Generate the radial profile centered at x=44 and y=37 (in pixels),
## up to a radial distance of 19 pixels, use the mean value.
$ astscript-radial-profile image.fits --center=44,37 --rmax=19

## Generate the radial profile centered at x=44 and y=37 (in pixels),
## up to a radial distance of 100 pixels, compute sigma clipped
## mean and standard deviation (sigclip-mean and sigclip-std) using
## 5 sigma and 0.1 tolerance (default is 3 sigma and 0.2 tolerance).
$ astscript-radial-profile image.fits --center=44,37 --rmax=100 \
   --sigmaclip=5,0.1 \
   --measure=sigclip-mean,sigclip-std

## Generate the radial profile centered at RA=20.53751695,
## DEC=0.9454292263, up to a radial distance of 88 pixels,
## axis ratio equal to 0.32, and position angle of 148 deg.
## Name the output table as 'radial-profile.fits'
$ astscript-radial-profile image.fits --mode=wcs \
   --center=20.53751695,0.9454292263 \
   --rmax=88 --axisratio=0.32 \
   --positionangle=148 -oradial-profile.fits

## Generate the radial profile centered at RA=40.062675270971,
## DEC=-8.1511992735126, up to a radial distance of 20 pixels,
## and calculate the SNR using the INPUT-NO-SKY and SKY-STD
## extensions of the NoiseChisel output file.
$ astscript-radial-profile image_detected.fits -hINPUT-NO-SKY \
   --mode=wcs --measure=sn \
   --center=40.062675270971,-8.1511992735126 \
   --rmax=20 --stdhdu=SKY_STD

## Generate the radial profile centered at RA=40.062675270971,
## DEC=-8.1511992735126, up to a radial distance of 20 pixels,
## and compute the SNR with a fixed value for std, std=10.
$ astscript-radial-profile image.fits -h1 --mode=wcs --rmax=20 \
   --center=40.062675270971,-8.1511992735126 \
   --measure=sn --instd=10

## Generate the radial profile centered at X=1201, Y=1201 pixels, up
## to a radial distance of 20 pixels and compute the median and the
## SNR using the first extension of sky-std.fits as the dataset for std
## values.
$ astscript-radial-profile image.fits -h1 --mode=img --rmax=20 \
   --center=1201,1201 --measure=median,sn \
   --instd=sky-std.fits
This installed script will read a FITS image and will use it as the basis for constructing the radial profile. The output radial profile is a table (FITS or plain-text) containing the radial distance from the center in the first row and the specified measurements in the other columns (mean, median, sigclip-mean, sigclip-median, etc.).

To measure the radial profile, this script needs to generate temporary files. All these temporary files will be created within the directory given to the \(--\text{tmpdir}\) option. When \(--\text{tmpdir}\) is not called, a temporary directory (with a name based on the inputs) will be created in the running directory. If the directory doesn’t exist at run-time, this script will create it. After the output is created, this script will delete the directory by default, unless you call the \(--\text{keeptmp}\) option.

With the default options, the script will generate a circular radial profile using the mean value and centered at the center of the image. In order to have more flexibility, several options are available to configure for the desired radial profile. In this sense, you can change the center position, the maximum radius, the axis ratio and the position angle (elliptical apertures are considered), the operator for obtaining the profiles, and others (described below).

**Debug your profile:** to debug your results, especially close to the center of your object, you can see the radial distance associated to every pixel in your input. To do this, use \(--\text{keeptmp}\) to keep the temporary files, and compare \texttt{crop.fits} (crop of your input image centered on your desired coordinate) with \texttt{apertures.fits} (radial distance of each pixel).

**Finding properties of your elliptical target:** you want to measure the radial profile of a galaxy, but don’t know its exact location, position angle or axis ratio. To obtain these values, you can use Section 7.2 [NoiseChisel], page 362, to detect signal in the image, feed it to Section 7.3 [Segment], page 381, to do basic segmentation, then use Section 7.4 [MakeCatalog], page 392, to measure the center (\(--\text{x}\) and \(--\text{y}\) in MakeCatalog), axis ratio (\(--\text{axisratio}\)) and position angle (\(--\text{positionangle}\)).

**Masking other sources:** The image of an astronomical object will usually have many other sources with your main target. A crude solution is to use sigma-clipped measurements for the profile. However, sigma-clipped measurements can easily be biased when the number of sources at each radial distance increases at larger distances. Therefore a robust solution is to mask all other detections within the image. You can use Section 7.2 [NoiseChisel], page 362, and Section 7.3 [Segment], page 381, to detect and segment the sources, then set all pixels that don’t belong to your target to blank using Section 6.2 [Arithmetic], page 272, (in particular, its \texttt{where} operator).

\texttt{-h STR}\n
\texttt{--hdu=STR}\n
The HDU/extension of the input image to use.
-o STR
--output=STR
Filename of measured radial profile. It can be either a FITS table, or plain-text table (determined from your given file name suffix).

-c FLT[,FLT[,...]]
--center=FLT[,FLT[,...]]
The central position of the radial profile. This option is used for placing the center of the profiles. This parameter is used in Section 6.1 [Crop], page 259, to center and crop the region. The positions along each dimension must be separated by a comma (,) and fractions are also acceptable. The number of values given to this option must be the same as the dimensions of the input dataset. The units of the coordinates are read based on the value to the --mode option, see below.

-O STR
--mode=STR
Interpret the center position of the object (values given to --center) in image or WCS coordinates. This option thus accepts only two values: img or wcs. By default, it is --mode=img.

-R FLT
--rmax=FLT
Maximum radius for the radial profile (in pixels). By default, the radial profile will be computed up to a radial distance equal to the maximum radius that fits into the image (assuming circular shape).

-Q FLT
--axisratio=FLT
The axis ratio of the apertures (minor axis divided by the major axis in a 2D ellipse). By default (when this option isn’t given), the radial profile will be circular (axis ratio of 1). This parameter is used as the option --qcol in the generation of the apertures with astmkprof.

-a FLT[,FLT[,...]]
--azimuth=FLT[,FLT[,...]]
Azimuthal angles range (in degrees, from 0 to 360) from the major axis. This parameter is used as the option --qcol=azimuth in the generation of the apertures with astmkprof. For example, if you want to obtain the radial profile only in between 55 and 78 degrees (taking as the reference the major axis), use --azimuth=55,78.

-p FLT
--positionangle=FLT
The position angle (in degrees) of the profiles relative to the first FITS axis (horizontal when viewed in SAO DS9). By default, it is --positionangle=0, which means that the semi-major axis of the profiles will be parallel to the first FITS axis.
The operator for measuring the values over each radial distance. The values given to this option will be directly passed to Section 7.4 [MakeCatalog], page 392. As a consequence, all MakeCatalog measurements like the magnitude, magnitude error, median, mean, signal-to-noise ratio (S/N), std, surface brightness, sigclip-mean, sigclip-number, etc. can be used here. For a full list of MakeCatalog's measurements, please run `astmkcatalog --help`. Multiple values can be given to this option, each separated by a comma. This option can also be called multiple times.

Some measurements by MakeCatalog require a per-pixel sky standard deviation (for example magnitude error or S/N). Therefore when asking for such measurements, use the `--instd` option (described below) to specify the per-pixel sky standard deviation over each pixel. For other measurements like the magnitude or surface brightness, MakeCatalog will need a Zeropoint, which you can set with the `--zeropoint` option.

For example, by setting `--measure=mean,sigclip-mean --measure=median`, the mean, sigma-clipped mean and median values will be computed. The output radial profile will have 4 columns in this order: radial distance, mean, sigma-clipped mean and median. By default (when this option isn't given), the mean of all pixels at each radial position will be computed.

Sigma clipping parameters: only relevant if sigma-clipping operators are requested by `--measure`. For more on sigma-clipping, see Section 7.1.3 [Sigma clipping], page 343. If given, the value to this option is directly passed to the `--sigmaclip` option of Section 7.4 [MakeCatalog], page 392, see Section 7.4.6.1 [MakeCatalog inputs and basic settings], page 409. By default (when this option isn't given), the default values within MakeCatalog will be used. To see the default value of this option in MakeCatalog, you can run this command:

```
$ astmkcatalog -P | grep " sigmaclip "
```

The Zeropoint of the input dataset. This is necessary when you request measurements like Magnitude, or Surface brightness.

Account for zero-valued pixels in the profile. By default, such pixels are not considered (when this script crops the necessary region of the image before generating the profile). The long format of this option is identical to a similarly named option in Crop (see Section 6.1.4 [Invoking Crop], page 263). When this option is called, it is passed directly to Crop, therefore the zero-valued pixels are not considered as blank and used in the profile creation.
-v INT
--oversample=INT
Oversample the input dataset to the fraction given to this option. Therefore if you set --rmax=20 for example and --oversample=5, your output will have 100 rows (without --oversample it will only have 20 rows). Unless the object is heavily undersampled (the pixels are larger than the actual object), this method provides a much more accurate result and there are sufficient number of pixels to get the profile accurately.

-u INT
--undersample=INT
Undersample the input dataset by the number given to this option. This option is for considering larger apertures than the original pixel size (aperture size is equal to 1 pixel). For example, if a radial profile computed by default has 100 different radii (apertures of 1 pixel width), by considering --undersample=2 the radial profile will be computed over apertures of 2 pixels, so the final radial profile will have 50 different radii. This option is good to measure over a larger number of pixels to improve the measurement.

-i FLT/STR
--instd=FLT/STR
Sky standard deviation as a single number (FLT) or as the filename (STR) containing the image with the std value for each pixel (the HDU within the file should be given to the --stdhdu option mentioned below). This is only necessary when the requested measurement (value given to --measure) by Make-Catalog needs the Standard deviation (for example the signal-to-noise ratio or magnitude error). If your measurements don’t require a standard deviation, it is best to ignore this option (because it will slow down the script).

-d INT/STR
--stdhdu=INT/STR
HDU/extension of the sky standard deviation image specified with --instd.

-t STR
--tmpdir=STR
Several intermediate files are necessary to obtain the radial profile. All of these temporal files are saved into a temporal directory. With this option, you can directly specify this directory. By default (when this option isn’t called), it will be built in the running directory and given an input-based name. If the directory doesn’t exist at run-time, this script will create it. Once the radial profile has been obtained, this directory is removed. You can disable the deletion of the temporary directory with the --keeptmp option.

-k
--keeptmp
Don’t delete the temporary directory (see description of --tmpdir above). This option is useful for debugging. For example, to check that the profiles generated for obtaining the radial profile have the desired center, shape and orientation.
10.3 SAO DS9 region files from table

Once your desired catalog (containing the positions of some objects) is created (for example with Section 7.4 [MakeCatalog], page 392, Section 7.5 [Match], page 429, or Section 5.3 [Table], page 230) it often happens that you want to see your selected objects on an image for a feeling of the spatial properties of your objects. For example you want to see their positions relative to each other.

In this section we describe a simple installed script that is provided within Gnuastro for converting your given columns to an SAO DS9 region file to help in this process. SAO DS9\(^2\) is one of the most common FITS image visualization tools in astronomy and is free software.

10.3.1 Invoking astscript-ds9-region

This installed script will read two positional columns within an input table and generate an SAO DS9 region file to visualize the position of the given objects over an image. For more on installed scripts please see (see Chapter 10 [Installed scripts], page 482). This script can be used with the following general template:

```bash
## Use the RA and DEC columns of 'table.fits' for the region file.
$ astscript-ds9-region table.fits --column=RA,DEC \  
   --output=ds9.reg

## Select objects with a magnitude between 18 to 20, and generate the  
## region file directly (through a pipe), each region with radius of  
## 0.5 arcseconds.
$ asttable table.fits --range=MAG,18:20 --column=RA,DEC \  
   | astscript-ds9-region --column=1,2 --radius=0.5

## With the first command, select objects with a magnitude of 25 to 26  
## as red regions in 'bright.reg'. With the second command, select  
## objects with a magnitude between 28 to 29 as a green region and  
## show both.
$ asttable cat.fits --range=MAG_F160W,25:26 -cRA,DEC \  
   | ./astscript-ds9-region -c1,2 --color=red -obright.reg
$ asttable cat.fits --range=MAG_F160W,28:29 -cRA,DEC \  
   | ./astscript-ds9-region -c1,2 --color=green \  
   | --command="ds9 image.fits -regions bright.reg"
```

The input can either be passed as a named file, or from standard input (a pipe). Only the --column option is mandatory (to specify the input table columns): two columns from the input table must be specified, either by name (recommended) or number. You can optionally also specify the region’s radius, width and color of the regions with the --radius, --width and --color options, otherwise default values will be used for these (described under each option).

The created region file will be written into the file name given to --output. When --output isn’t called, the default name of ds9.reg will be used (in the running directory). If the file exists before calling this script, it will be overwritten, unless you pass

\(^2\) http://ds9.si.edu
Chapter 10: Installed scripts

the --dontdelete option. Optionally you can also use the --command option to give the full command that should be run to execute SAO DS9 (see example above and description below). In this mode, the created region file will be deleted once DS9 is closed (unless you pass the --dontdelete option). A full description of each option is given below.

-h INT/STR
--hdru INT/STR
The HDU of the input table when a named FITS file is given as input. The HDU (or extension) can be either a name or number (counting from zero). For more on this option, see Section 4.1.2.1 [Input/Output options], page 157.

-c STR,STR
--column=STR,STR
Identifiers of the two positional columns to use in the DS9 region file from the table. They can either be in WCS (RA and Dec) or image (pixel) coordinates. The mode can be specified with the --mode option, described below.

-n STR
--namecol=STR
The column containing the name (or label) of each region. The type of the column (numeric or a character-based string) is irrelevant: you can use both types of columns as a name or label for the region. This feature is useful when you need to recognize each region with a certain ID or property (for example magnitude or redshift).

-m wcs|img
--mode=wcs|org
The coordinate system of the positional columns (can be either --mode=wcs and --mode=img). In the WCS mode, the values within the columns are interpreted to be RA and Dec. In the image mode, they are interpreted to be pixel X and Y positions. This option also affects the interpretation of the value given to --radius. When this option isn’t explicitly given, the columns are assumed to be in WCS mode.

-C STR
--color=STR
The color to use for created regions. These will be directly interpreted by SAO DS9 when it wants to open the region file so it must be recognizable by SAO DS9. As of SAO DS9 8.2, the recognized color names are black, white, red, green, blue, cyan, magenta and yellow. The default color (when this option is not called) is green.

-w INT
--width=INT
The line width of the regions. These will be directly interpreted by SAO DS9 when it wants to open the region file so it must be recognizable by SAO DS9. The default value is 1.
Chapter 10: Installed scripts

- \texttt{r FLT}

\texttt{--radius=FLT}

The radius of all the regions. In WCS mode, the radius is assumed to be in arc-seconds, in image mode, it is in pixel units. If this option is not explicitly given, in WCS mode the default radius is 1 arc-seconds and in image mode it is 3 pixels.

\texttt{--dontdelete}

If the output file name exists, abort the program and don’t over-write the contents of the file. This option is thus good if you want to avoid accidentally writing over an important file. Also, don’t delete the created region file when \texttt{--command} is given (by default, when \texttt{--command} is given, the created region file will be deleted after SAO DS9 closes).

\texttt{-o STR}

\texttt{--output=STR}

Write the created SAO DS9 region file into the name given to this option. If not explicitly given on the command-line, a default name of \texttt{ds9.reg} will be used. If the file already exists, it will be over-written, you can avoid the deletion (or over-writing) of an existing file with the \texttt{--dontdelete}.

\texttt{--command="STR"}

After creating the region file, run the string given to this option as a command-line command. The SAO DS9 region command will be appended to the end of the given command. Because the command will mostly likely contain white-space characters it is recommended to put the given string in double quotations. For example, let’s assume \texttt{--command="ds9 image.fits -zscale"}. After making the region file (assuming it is called \texttt{ds9.reg}), the following command will be executed:

\begin{verbatim}
  ds9 image.fits -zscale -regions ds9.reg
\end{verbatim}

You can customize all aspects of SAO DS9 with its command-line options, therefore the value of this option can be as long and complicated as you like. For example if you also want the image to fit into the window, this option will be: \texttt{--command="ds9 image.fits -zscale -zoom to fit"}. You can see the SAO DS9 command-line descriptions by clicking on the “Help” menu and selecting “Reference Manual”. In the opened window, click on “Command Line Options”.

10.4 Viewing FITS file contents with DS9 or TOPCAT

The FITS definition allows for multiple extensions (or HDUs) inside one FITS file. Each HDU can have a completely independent dataset inside of it. One HDU can be a table, another can be an image and another can be another independent image. For example each image HDU can be one CCD of a multi-CCD camera, or in processed images one can be the deep science image and the next can be its weight map, alternatively, one HDU can be an image, and another can be the catalog/table of objects within it.

The most common software for viewing FITS images is SAO DS9 (see Section B.1 [SAO DS9], page 702) and for plotting tables, TOPCAT is the most commonly used tool in
astronomy (see Section B.2 [TOPCAT], page 703). After installing them (as described in
the respective appendix linked in the previous sentence), you can open any number of FITS
images or tables with DS9 or TOPCAT with the commands below:

$ ds9 image-a.fits image-b.fits
$ topcat table-a.fits table-b.fits

But usually the default mode is not enough. For example in DS9, the window can be
too small (not covering the height of your monitor), you probably want to match and lock
multiple images, you have a favorite color map that you prefer to use, or you may want to
open a multi-extension FITS file as a cube.

Using the simple commands above, you need to manually do all these in the DS9 window
once it opens and this can take several tens of seconds (which is enough to distract you
from what you wanted to inspect). For example if you have a multi-extension file containing
2D images, one way to load and switch between each 2D extension is to take the following
steps in the SAO DS9 window: “File”→“Open Other”→“Open Multi Ext Cube” and then
choose the Multi extension FITS file in your computer’s file structure.

The method above is a little tedious to do every time you want view a multi-extension
FITS file. A different series of steps is also necessary if you the extensions are 3D data
cubes (since they are already cubes, and should be opened as multi-frame). Furthermore,
if you have multiple images and want to “match” and “lock” them (so when you zoom-in
to one, all get zoomed-in) you will need several other sequence of menus and clicks.

Fortunately SAO DS9 also provides command-line options that you can use to specify
a particular behavior before/after opening a file. One of those options is -mecube which
opens a FITS image as a multi-extension data cube (treating each 2D extension as a slice
in a 3D cube). This allows you to flip through the extensions easily while keeping all the
settings similar. Just to avoid confusion, note that SAO DS9 does not follow the GNU style
of separating long and short options as explained in Section 4.1.1 [Arguments and options],
page 153. In the GNU style, this ‘long’ (multi-character) option should have been called
like --mecube, but SAO DS9 follows its own conventions.

For example try running $ds9 -mecube foo.fits to see the effect (for example on the
output of Section 7.2 [NoiseChisel], page 362). If the file has multiple extensions, a small
window will also be opened along with the main DS9 window. This small window allows
you to slide through the image extensions of foo.fits. If foo.fits only consists of one
extension, then SAO DS9 will open as usual.

On the other hand, for visualizing the contents of tables (that are also commonly stored
in the FITS format), you need to call a different software (most commonly, people use
TOPCAT, see Section B.2 [TOPCAT], page 703). And to make things more inconvenient,
by default both of these are only installed as command-line software, so while you are
navigating in your GUI, you need to open a terminal there, and run these commands. All
of the issues above are the founding purpose of the installed script that is introduced in
Section 10.4.1 [Invoking astscript-fits-view], page 495.

10.4.1 Invoking astscript-fits-view

Given any number of FITS files, this script will either open SAO DS9 (for images or cubes)
or TOPCAT (for tables) to visualize their contents in a graphic user interface (GUI). For
more on installed scripts please see (see Chapter 10 [Installed scripts], page 482). This script can be used with the following general template:

\[
\text{
$\text{astscript-fits-view \ [OPTION]} \ \text{input.fits \ [input-b.fits \ ...]$
}
\]

One line examples

## Call TOPCAT to load all the input FITS tables.
$\text{astscript-fits-view table-*.fits}$

## Call SAO DS9 to open all the input FITS images.
$\text{astscript-fits-view image-*.fits}$

This script will use Gnuastro’s Section 5.1 [Fits], page 194, program to see if the file is a table or image. If the first input file contains an image HDU, then the sequence of files will be given to Section B.1 [SAO DS9], page 702. Otherwise, the input(s) will be given to Section B.2 [TOPCAT], page 703, to visualize (plot) as tables. When opening DS9 it will also inspect the dimensionality of the first image HDU of the first input and open it slightly differently when the input is 2D or 3D:

2D  

DS9’s \(-\text{mecube}\) will be used to open all the 2D extensions of each input file as a “Multi-extension cube”. A “Cube” window will also be opened with DS9 that can be used to slide/flip through each extensions. When multiple files are given, each file will be in one “frame”.

3D  

DS9’s \(-\text{multiframe}\) option will be used to open all the extensions in a separate “frame” (since each input is already a 3D cube, the \(-\text{mecube}\) option can be confusing). To flip through the extensions (while keeping the slice fixed), click the “frame” button on the top row of buttons, then use the last four buttons of the bottom row ("first", "previous", "next" and "last") to change between the extensions. If multiple files are given, there will be a separate frame for each HDU of each input (each HDU’s name or number will be put in square brackets after its name).

Double-clicking on FITS file to open DS9 or TOPCAT: for those graphic user interface (GUI) that follow the freedesktop.org standards (including GNOME, KDS Plasma, or Xfce) Gnuastro installs a \text{fits-view.desktop} file to instruct your GUI to call this script for opening FITS files when you click on them. To activate this feature take the following steps:

1. Run the following command, while replacing \text{PREFIX}. If you don’t know what to put in \text{PREFIX}, run \text{which astfits} on the command-line, and extract \text{PREFIX} from the output (the string before /bin/astfits). For more, see Section 3.3.1.2 [Installation directory], page 139.

\[
\text{ln -sf \text{PREFIX/share/gnuastro/astscript-fits-view.desktop} \ }
\text{~/.local/share/applications/}
\]

2. Right-click on a FITS file, and choose these items in order (based on GNOME, may be different in KDE or Xfce): “Open with other application”→“View all applications”→“astscript-fits-view”.


This script takes the following options

```
-h STR
--hdu=STR
```

The HDU (extension) of the input dataset to display. The value can be the HDU name or number (the first HDU is counted from 0).

```
-p STR
--prefix=STR
```

Directory to search for SAO DS9 or TOPCAT’s executables (assumed to be `ds9` and `topcat`). If not called they will be assumed to be present in your `PATH` (see Section 3.3.1.2 [Installation directory], page 139). If you don’t have them already installed, their installation directories are available in Section B.1 [SAO DS9], page 702, and Section B.2 [TOPCAT], page 703, (they can be installed in non-system-wide locations that don’t require administrator/root permissions).

```
-s STR
--ds9scale=STR
```

The string to give to DS9’s `-scale` option. You can use this option to use a different scaling. The Fits-view script will place `-scale` before your given string when calling DS9. If you don’t call this option, the default behavior is to cal DS9 with: `-scale mode zscale` or `--ds9scale="mode zscale"` when using this script.

The Fits-view script has the following aliases to simplify the calling of this option (and avoid the double-quotations and `mode` in the example above):

```
zscale       or `--ds9scale=zscale` equivalent to `--ds9scale="mode zscale"`
minmax       or `--ds9scale=minmax` equivalent to `--ds9scale="mode minmax"`
```

```
c=FLT,FLT
--ds9center=FLT,FLT
```

The central coordinate for DS9’s view of the FITS image after it opens. This is equivalent to the “Pan” button in DS9. The nature of the coordinates will be determined by the `--ds9mode` option that is described below.

```
-o img/wcs
--ds9mode=img/wcs
```

The coordinate system (or mode) to interpret the values given to `--ds9center`. This can either be `img` (or DS9’s “Image” coordinates) or `wcs` (or DS9’s “wcs fk5” coordinates).

```
g INTxINT
--ds9geometry=INTxINT
```

The initial DS9 window geometry (value to DS9’s `-geometry` option).

10.5 PSF construction and subtraction

The point spread function (PSF) describes how the light of a point-like source is affected by several optical scattering effects (atmosphere, telescope, instrument, etc.). Since the light of
all astrophysical sources undergoes all these effects, characterizing the PSF is key in astronomical analysis (for small and large objects). Consequently, having a good characterization of the PSF is fundamental to any analysis.

In some situations\(^3\) a parametric (analytical) model is sufficient for the PSF (Gaussian, Moffat, etc., see Section 8.1.1.2 [Point spread function], page 442). However, once you are interested in objects that are larger than a handful of pixels, it is almost impossible to find an analytic function to adequately characterize the PSF. Therefore, it is necessary to obtain an empirical (non-parametric) and extended PSF. In this section we describe a set of installed scripts in Gnuastro that will let you construct the non-parametric PSF using point-like sources. They allow you to derive the PSF from the same astronomical images that the science is derived from (without assuming any analytical function).

The scripts are based on the concepts described in Infante-Sainz et al. (2020, https://arxiv.org/abs/1911.01430). But to be complete, we first give a summary of the logic and overview of their combined usage in Section 10.5.1 [Overview of the PSF scripts], page 498. Furthermore, before going into the technical details of each script, we encourage you to go through the tutorial that is devoted to this at Section 2.4 [Building the extended PSF], page 97. The tutorial uses a real dataset and includes all the logic and reasoning behind every step of the usage in every installed script.

10.5.1 Overview of the PSF scripts

To obtain an extended and non-parametric PSF, several steps are necessary and we will go through them here. The fundamental ideas of the following methodology are thoroughly described in Infante-Sainz et al. (2020, https://arxiv.org/abs/1911.01430). A full tutorial is also available in Section 2.4 [Building the extended PSF], page 97. The tutorial will go through the full process on a pre-selected dataset, but will describe the logic behind every step in away that can easily be modified/generalized to other datasets.

This section is basically just a summary of that tutorial. We could have put all these steps into one large program (installed script), however this would introduce several problems. The most prominent of these problems are:

- The command would require many options, making it very complex to run every time.
- You usually have many stars in an image, and many of the steps can be optimized or parallelized depending on the particular analysis scenario. Predicting all the possible optimizations for all the possible usage scenarios would make the code extremely complex (filled with many unforeseen bugs!).

Therefore, following the modularity principle of software engineering, after several years of working on this, we have broken the full job into the smallest number of independent steps as separate scripts. All scripts are independent of each other, meaning this that you are free to use all of them as you wish (for example only some of them, using another program for a certain step, using them for other purposes, or running independent parts in parallel).

For constructing the PSF from your dataset, the first step is to obtain a catalog of stars within it (you can’t use galaxies to build the PSF!). But you can’t blindly use all the stars

---

\(^3\) An example scenario where a parametric PSF may be enough: you are only interested in very small, high redshift objects that only extended a handful of pixels.
either! For example we don’t want contamination from other bright, and nearby objects. The first script below is therefore designed for selecting only good star candidates in your image. It will use different criteria, for example good parallax (where available, to avoid confusion with galaxies), not being near to bright stars, axis ratio, etc. For more on this script, see Section 10.5.2 [Invoking astscript-psf-select-stars], page 500.

Once the catalog of stars is constructed, another script is in charge of making appropriate stamps of the stars. Each stamp is a cropped image of the star with the desired size, normalization of the flux, and mask of the contaminant objects. For more on this script, see Section 10.5.3 [Invoking astscript-psf-stamp], page 503. After obtaining a set of star stamps, they can be stacked for obtaining the combined PSF from many stars (for example with Section 6.2.3.5 [Stacking operators], page 283).

In the combined PSF, the masked background objects of each star’s image will be covered and the signal-to-noise ratio will increase, giving a very nice view of the “clean” PSF. However, it is usually necessary to obtain different regions of the same PSF from different stars. For example, to construct the far outer wings of the PSF, it is necessary to consider very bright stars. However, these stars will be saturated in the most inner part, and immediately outside of the saturation level, they will be deformed due to non-linearity effects. Consequently, fainter stars are necessary for the inner regions.

Therefore, you need to repeat the steps above for certain stars (in a certain magnitude range) to obtain the PSF in certain radial ranges. For example, in Infante-Sainz et al. (2020, https://arxiv.org/abs/1911.01430), the final PSF was constructed from three regions (and thus, using stars from three ranges in magnitude). In other cases, we even needed four groups of stars! But in the example dataset from the tutorial, only two groups are necessary (see Section 2.4 [Building the extended PSF], page 97).

Once clean stacks of different parts of the PSF have been constructed through the steps above, it is therefore necessary to blend them all into one. This is done by finding a common radial region in both, and scaling the inner region by a factor to add with the outer region. This is not trivial, therefore, a third script is in charge of it, see Section 10.5.4 [Invoking astscript-psf-unite], page 506.

Having constructed the PSF as described above (or by any other procedure), it can be scaled to the brightness of the various stars in the image to get subtracted (and thus remove the extended/bright wings; better showing the background objects of interest). Note that the absolute flux of a PSF is meaningless (and in fact, it is usually normalized to have a total sum of unity!), so it should be scaled. We therefore have another script that will calculate the scale (multiplication) factor of the PSF for each star. For more on the scaling script, see Section 10.5.5 [Invoking astscript-psf-scale-factor], page 508.

Once the flux factor has been computed, a final script is in charge of placing the scaled PSF over the proper location in the image, and subtracting it. It is also possible to only obtain the modeled star by the PSF. For more on the scaling and positioning script, see Section 10.5.6 [Invoking astscript-psf-subtract], page 510.

As mentioned above, in the following sections, each script has its own documentation and list of options for very detailed customization (if necessary). But if you are new to these scripts, before continuing, we recommend that you do the tutorial Section 2.4 [Building the extended PSF], page 97. Just don’t forget to run every command, and try to tweak its steps based on the logic to nicely understand it.
10.5.2 Invoking astscript-psf-select-stars

This installed script will select good star candidates for constructing a PSF. It will consider stars within a given range of brightness without nearby contaminant objects. To do that, it allows to the user to specify different options described here. A complete tutorial is available to show the operation of this script as a modular component to extract the PSF of a dataset: Section 2.4 [Building the extended PSF], page 97. The executable name is astscript-psf-select-stars, with the following general template:

$ astscript-psf-select-stars [OPTION...] FITS-file

Examples:

## Select all stars within 'image.fits' with magnitude in range
## of 6 to 10; only keeping those that are less than 0.02 degrees
## from other nearby stars.
$ astscript-psf-select-stars image.fits \
   --magnituderange=6,10 --mindistdeg=0.02

The input of this script is an image, and the output is a catalog of stars with brightness in the requested range of magnitudes (provided with --magnituderange). The output catalog will also only contain stars that are sufficiently distant (--mindistdeg) from all other brighter, and some fainter stars. It is possible to consider different datasets with the option --dataset (by default, Gaia eDR3 dataset is considered) All stars that are --faintmagdiff fainter than the faintest limit will also be accounted for, when selecting good stars. The --magnituderange, and --mindistdeg are mandatory: if not specified the code will abort.

The output of this script is a file whose name can be specified with the (optional) --output option. If not given, an automatically generated name will be used for the output. A full description of each option is given below.

-h STR/INT
   --hdu=STR/INT
   The HDU/extension of the input image to use.

-S STR
   --segmented=STR
   Optional segmentation file obtained by Section 7.3 [Segment], page 381. It should have two extensions (CLUMPS and OBJECTS). If given, a catalog of CLUMPS will be computed and matched with the Gaia catalog to reject those objects that are too elliptical (see --minaxisratio). The matching will occur on an aperture (in degrees) specified by --matchaperturedeg.

-a FLT
   --matchaperturedeg=FLT
   This option determines the aperture (in degrees) for matching the catalog from gaia with the clumps catalog that is produced by the segmentation image given to --segmented. The default value is 10 arc-seconds.
-c STR
   --catalog=STR
   Optional reference catalog to use for selecting stars (instead of querying an external catalog like Gaia). When this option is given, --dataset (described below) will be ignored and no internet connection will be necessary.

-d STR
   --dataset=STR
   Optional dataset to query (see Section 5.4 [Query], page 249). It should contain the database and dataset entries to Query. Its value will be immediately given to astquery. By default, its value is gaia --dataset=edr3 (so it connects to the Gaia database and requests the early data release 3).

   It is possible to specify a different dataset from which the catalog is downloaded. In that case, the necessary column names may also differ, so you also have to set --racolumn, --deccolumn and --field. See their description for more.

-r STR
   --racolumn=STR
   The name of the column containing the Right Ascension (RA) in the requested dataset (--dataset). If the user does not determine this option, the default value is assumed to be ra.

-d STR
   --deccolumn=STR
   The name of the column containing the Declination (Dec) in the requested dataset (--dataset). If the user does not determine this option, the default value is assumed to be dec.

-f STR
   --field=STR
   The name of the column containing the magnitude in the requested dataset (--dataset). The output will only contain stars that have a value in this column, between the values given to --magnituderange (see below). By default, the value of this option is phot_g_mean_mag (that corresponds to the name of the magnitude of the G-band in the Gaia catalog).

-m FLT,FLT
   --magnituderange=FLT,FLT
   The acceptable range of values for the column in --field. This option is mandatory and no default value is assumed.

-p STR,STR
   --parallaxanderrorcolumn=STR,STR
   With this option the user can provide the parallax and parallax error column names in the requested dataset. When given, the output will only contain stars for which the parallax value is smaller than three times the parallax error. If the user does not provide this option, the script will not use parallax information for selecting the stars. In the case of Gaia, if you want to use parallax to further limit the good stars, you can pass parallax,parallax_error.
-D FLT
--mindistdeg=FLT
Stars with nearby bright stars closer than this distance are rejected. The default value is 1 arc minute. For fainter stars (when constructing the center of the PSF), you should decrease the value.

-b INT
--brightmag=INT
The brightest star magnitude to avoid (should be brighter than the brightest of --magnituderange). The basic idea is this: if a user asks for stars with magnitude 6 to 10 and one of those stars is near a magnitude 3 star, that star (with a magnitude of 6 to 10) should be rejected because it is contaminated. But since the catalog is constrained to stars of magnitudes 6-10, the star with magnitude 3 is not present and cannot be compared with! Therefore, when considering proximity to nearby stars, it is important to use a larger magnitude range than the user’s requested magnitude range for good stars. The acceptable proximity is defined by --mindistdeg.

With this option, you specify the brightest limit for the proximity check. The default value is a magnitude of −10, so you’ll rarely need to change or customize this option!

The faint limit of the proximity check is specified by --faintmagdiff. As the name suggests, this is a “diff” or relative value. The default value is 4. Therefore if the user wants to build the PSF with stars in the magnitude range of 6 to 10, the faintest stars used for the proximity check will have a magnitude of 14: 10 + 4. In summary, by default, the proximity check will be done with stars in the magnitude range −10 to 14.

-F INT
--faintmagdiff
The magnitude difference of the faintest star used for proximity checks to the faintest limit of --magnituderange. For more, see the description of --brightmag.

-Q FLT
--minaxisratio=FLT
Minimum acceptable axis ratio for the selected stars. In other words, only stars with axis ratio between --minaxisratio to 1.0 will be selected. Default value is --minaxisratio=0.9. Recall that the axis ratio is only used when you also give a segmented image with --segmented.

-t
--tmpdir
Directory to keep temporary files during the execution of the script. If the directory doesn’t exist at run-time, this script will create it. By default, upon completion of the script, this directory will be deleted. However, if you would like to keep the intermediate files, you can use the --keeptmp option.

-k
--keeptmp
Do not remove the temporary directory (see description of --keeptmp). This option is useful for debugging and checking the outputs of internal steps.
-o STR
--output=STR

The output name of the final catalog containing good stars.

### 10.5.3 Invoking astscript-psf-stamp

This installed script will generate a stamp of fixed size, centered at the provided coordinates (performing sub-pixel regridding if necessary) and normalized at a certain normalization radius. Optionally, it will also mask all the other background sources. A complete tutorial is available to show the operation of this script as a modular component to extract the PSF of a dataset: Section 2.4 [Building the extended PSF], page 97. The executable name is `astscript-psf-stamp`, with the following general template:

```bash
$ astscript-psf-stamp [OPTION...] FITS-file
```

Examples:

```bash
## Make a stamp around (x,y)=(53,69) of width=151 pixels.
## Normalize the stamp within the radii 20 and 30 pixels.
$ astscript-psf-stamp image.fits --mode=img \  
  --center=53,69 --stampwidth=151 --normradii=20,30 \  
  --output=stamp.fits

## Iterate over a catalog with positions of stars that are
## in the input image. Use WCS coordinates.
$ asttable catalog.fits | while read -r ra dec mag; do \  
  astscript-psf-stamp image.fits \  
  --mode=wcs \  
  --stampwidth=150 \  
  --center=$ra,$dec \  
  --normradii=20,30 \  
  --output=stamp-"$ra"-"$dec".fits; done
```

The input is an image from which the stamp of the stars are constructed. There are mandatory options that the user has to specify: `--mode`, `--center`, `--stampwidth` and `--normradii`. The output will be an image with a size specified by `--stampwidth`, centered at the position specified by the option `--center`. The output will be “normalized” by the value computed within the ring around the center (at a radial distance between the two radii specified by the option `--normradii`).

If the given coordinate is not within the image at all (such that the full requested stamp is outside the input image), then this script will abort with a message and a non-zero return value. If all the pixels within the normalization radii are NaN, the output stamp will be fully NaN. This can happen when the stamp is only partially out of the original image field of view, or there is a bad pixel region over the star. A fully NaN image can safely be used with the Stacking operators of Arithmetic (see Section 6.2.3.5 [Stacking operators], page 283): in effect it will be ignored.

More options are available with the goal of obtaining better stamps. A full description of each option is given below.
-h STR
   --hdu=STR
   The HDU/extension of the input image to use.

-O STR
   --mode=STR
   Interpret the center position of the object (values given to --center) in image
   or WCS coordinates. This option thus accepts only two values: img or wcs.

-c FLT,FLT
   --center=FLT,FLT
   The central position of the object. This option is used for placing the center
   of the stamp. This parameter is used in Section 6.1 [Crop], page 259, to center
   and crop the image. The positions along each dimension must be separated by
   a comma (,). The units of the coordinates are read based on the value to the
   --mode option, see above.

-W INT
   --stampwidth=INT
   Size (width) of the output image stamp in pixels. The size of the output image
   will be always an odd number of pixels. As a consequence, if the user specify
   an even number, the final size will be the specified size plus 1 pixel. This is
   necessary to place the specified coordinate (given to --center) in the center
   of the central pixel. This is very important (and necessary) in the case of the
   centers of stars, therefore a sub-pixel translation will be performed internally
   to ensure this.

-n FLT,FLT
   --normradii=FLT,FLT
   Minimum and maximum radius of ring to normalize the image. This option
   takes two values, separated by a comma (,). The first value is the inner radius,
   the second is the outer radius.

-S STR
   --segment=STR
   Optional filename of a segmentation image from Segment’s output (must con-
   tain the CLUMPS and OBJECT HDUs). For more on the definition of “objects”
   and “clumps”, see Section 7.3 [Segment], page 381. If given, Segment’s output
   is used to mask all background sources from the large foreground object (a
   bright star):
   
   - Objects that are not the central object.
   - Clumps (within the central object) that are not the central clump.

The result is that all objects and clumps that contaminate the central source
are masked, while the diffuse flux of the central object remains. The size of
the region that is used for identifying the central clump/object label can be
specified by the option --corewidth, see below.
-w INT
--corewidth=INT
Width of the region that is used to compute the central clump/object label over
the segmentation image --segment, with the goal of not masking them. It is
rarely necessary to change this option.

If a segmentation image is provided, then it is necessary to not mask the central
object+clump (see description of --segment). To do that, the clump and object
images are cropped around the coordinate given to --center, but on the two
labeled images. Assuming a single value exists within the crop, the IDs of the
respective object and clump are found. Therefore it is necessary for this option
to be small (the program will abort if more than one label exists within it).

-N STR
--normop=STR
The operator for measuring the values within the ring defined by the option
--normradii. The operator given to this option will be directly passed to the
radial profile script astscript-radial-profile, see Section 10.2 [Generate
radial profile], page 486. As a consequence, all MakeCatalog measurements
(median, mean, sigclip-mean, sigclip-number, etc.) can be used here. For a full
list of MakeCatalog’s measurements, please run astmkcatalog --help.

-Q FLT
--axisratio=FLT
The axis ratio of the radial profiles for computing the normalization value. By
default (when this option isn’t given), the radial profile will be circular (axis
ratio of 1). This parameter is used directly in the astscript-radial-profile
script.

-p FLT
--positionangle=FLT
The position angle (in degrees) of the profiles relative to the first FITS axis
(horizontal when viewed in SAO DS9). By default, it is --positionangle=0,
which means that the semi-major axis of the profiles will be parallel to the first
FITS axis. This parameter is used directly in the astscript-radial-profile
script.

-s FLT,FLT
--sigmaclip=FLT,FLT
Sigma clipping parameters: only relevant if sigma-clipping operators are re-
quested by --normop. For more on sigma-clipping, see Section 7.1.3 [Sigma
clipping], page 343.

-t
--tmpdir
Directory to keep temporary files during the execution of the script. If the
directory doesn’t exist at run-time, this script will create it. By default, upon
completion of the script, this directory will be deleted. However, if you would
like to keep the intermediate files, you can use the --keeptmp option.
-k
--keeptmp
Do not remove the temporary directory (see description of --keeptmp). This option is useful for debugging and checking the outputs of internal steps.

-o STR
--output=STR
Filename of stamp image. By default the name of the stamp will be a combination of the input image name, the name of the script, and the coordinates of the center. For example, if the input image is named image.fits and the center is --center=33,78, then the output name will be: image_stamp_33_78.fits. The main reason of setting this name is to have an unique name for each stamp by default.

10.5.4 Invoking astscript-psf-unite
This installed script will join two PSF images at a given radius. This operation is commonly used when merging (uniting) the inner and outer parts of the PSF. A complete tutorial is available to show the operation of this script as a modular component to extract the PSF of a dataset: Section 2.4 [Building the extended PSF], page 97. The executable name is astscript-psf-unite, with the following general template:

$ astscript-psf-unite [OPTION...] FITS-file

Examples:

## Multiply inner.fits by 3 and put it in the center (within a radius of 25 pixels) of outer.fits. The core goes up to a radius of 25 pixels.
$ astscript-psf-unite outer.fits \\
   --core=inner.fits --scale=3 \\
   --radius=25 --output=joined.fits

## Same example than the above, but considering an ellipse (instead of a circle).
$ astscript-psf-unite outer.fits \\
   --core=inner.fits --scale=3 \\
   --radius=25 --axisratio=0.5 \\
   --positionangle=40 --output=joined.fits

The junction is done by considering the input image as the outer part. The central part is specified by FITS image given to --inner and it is multiplied by the factor --scale. All pixels within --radius (in pixels) of the center of the outer part are then replaced with the inner image.

The scale factor to multiply with the inner part has to be explicitly provided (see the description of --scale below). Note that this script assumes that PSF is centered in both images. More options are available with the goal of obtaining a good junction. A full description of each option is given below.

-h STR
--hdu=STR
The HDU/extension of the input image to use.
Chapter 10: Installed scripts

-i STR
--inner=STR
Filename of the inner PSF. This image is considered to be the central part of the PSF. It will be cropped at the radius specified by the option --radius, and multiplied by the factor specified by --scale. After that, it will be appended to the outer part (input image).

-I STR
--innerhdu=STR
The HDU/extension of the inner PSF (option --inner).

-f FLT
--scale=FLT
Factor by which the inner part (--inner) is multiplied. This factor is necessary to put the two different parts of the PSF at the same flux level. A convenient way of obtaining this value is by using the script astscript-model-scale-factor, see Section 10.5.5 [Invoking astscript-psf-scale-factor], page 508. There is also a full tutorial on using all the astscript-psf-* installed scripts together, see Section 2.4 [Building the extended PSF], page 97. We recommend doing that tutorial before starting to work on your own datasets.

-r FLT
--radius=FLT
Radius (in pixels) at which the junction of the images is done. All pixels in the outer image within this radius (from its center) will be replaced with the pixels of the inner image (that has been scaled). By default, a circle is assumed for the shape of the inner region, but this can be tweaked with --axisratio and --positionangle (see below).

-Q FLT
--axisratio=FLT
Axis ratio of ellipse to define the inner region. By default this option has a value of 1.0, so all central pixels (of the outer image) within a circle of radius --radius are replaced with the scaled inner image pixels. With this option, you can customize the shape of pixels to take from the inner and outer profiles. For a PSF, it will usually not be necessary to change this option: even if the PSF is non-circular, the inner and outer parts will both have the same ellipticity. So if the scale factor is chosen accurately, using a circle to select which pixels from the inner image to use in the outer image will be irrelevant.

-p FLT
--positionangle=FLT
Position angle of the ellipse (in degrees) to define which central pixels of the outer image to replace with the scaled inner image. Similar to --axisratio (see above).

-t
--tmpdir
Directory to keep temporary files during the execution of the script. If the directory doesn’t exist at run-time, this script will create it. By default, upon completion of the script, this directory will be deleted. However, if you would like to keep the intermediate files, you can use the --keeptmp option.
-k
--keeptmp

Do not remove the temporary directory (see description of --keeptmp). This option is useful for debugging and checking the outputs of internal steps.

### 10.5.5 Invoking astscript-psf-scale-factor

This installed script will compute the multiplicative factor (scale) that is necessary to match the PSF to a given star. The match in flux is done within a ring of pixels. It can also be used to compute the scale factor to multiply the inner part of the PSF with the outer part during the creation of a PSF. A complete tutorial is available to show the operation of this script as a modular component to extract the PSF of a dataset: Section 2.4 [Building the extended PSF], page 97. The executable name is astscript-psf-scale-factor, with the following general template:

```
$ astscript-psf-scale-factor [OPTION...] FITS-file
```

Examples:

```bash
## Compute the scale factor for the object at (x,y)=(53,69) for
## the PSF (psf.fits). Compute it in the ring 20-30 pixels.
$ astscript-psf-scale-factor image.fits --mode=img \  
  --center=53,69 --normradii=20,30 --psf=psf.fits

## Iterate over a catalog with RA,Dec positions of stars that are in
## the input image to compute their scale factors.
$ asttable catalog.fits | while read -r ra dec mag; do \  
  astscript-psf-scale-factor image.fits \  
  --mode=wcs \  
  --psf=psf.fits \  
  --center=$ra,$dec --quiet \  
  --normradii=20,30 > scale-"$ra"-"$dec".txt; done
```

The input should be an image containing the star that you want to match in flux with the PSF. The output will be a single number that is printed on the command-line. That number is the multiplicative factor to scale the PSF image (given to --psf) to match in flux with the given star (which is located in --center coordinate of the input image). The scale factor will be calculated within the ring of pixels specified by the option --normradii.

All the pixels within this ring will be separated from both the PSF and input images. For the input image, around the selected coordinate; while masking all other sources (see --segment). The finally selected pixels of the input image will then be divided by those of the PSF image. This gives us an image containing one scale factor per pixel. The finally reported value is the sigma-clipped median of all the scale factors in the finally-used pixels. To fully understand the process on first usage, we recommend that you run this script with --keeptmp and inspect the files inside of the temporary directory.

The most common use-cases of this scale factor are:

1. To find the factor for joining two different parts of the same PSF, see Section 10.5.4 [Invoking astscript-psf-unite], page 506.
2. When modeling a star in order to subtract it using the PSF, see Section 10.5.6 [Invoking 
astscript-psf-subtract], page 510.

For a full tutorial on how to use this script along with the other astscript-psf-* 
scripts in Gnuastro, please see Section 2.4 [Building the extended PSF], page 97. To allow 
full customizability, the following options are available with this script.

- `h STR` 
  `--hdu=STR` 
  The HDU/extension of the input image to use.

- `p STR` 
  `--psf=STR` 
  Filename of the PSF image. The PSF is assumed to be centered in this image.

- `P STR` 
  `--psfhdu=STR` 
  The HDU/extension of the PSF image.

- `c FLT,FLT` 
  `--center=FLT,FLT` 
  The central position of the object to scale with the PSF. This parameter is 
passed to Gnuastro’s Crop program make a crop for further processing (see 
Section 6.1 [Crop], page 259). The positions along each dimension must be 
separated by a comma (,). The units of the coordinates are interpreted based 
on the value to the `--mode` option (see below).

- `O STR` 
  `--mode=STR` 
  Interpret the center position of the object (values given to `--center`) in image 
or WCS coordinates. This option thus accepts only two values: `img` or `wcs`.

- `n INT,INT` 
  `--normradii=INT,INT` 
  Inner (inclusive) and outer (exclusive) radii (in units of pixels) around the 
central position in which the scale factor is computed. The option takes two 
values separated by a comma (,). The first value is the inner radius, the second 
is the outer radius. These two radii define a ring of pixels around the center 
that is used for obtaining the scale factor value.

- `W INT` 
  `--stampwidth=INT` 
  Size (width) of the image stamp in pixels. This is an intermediate product 
computed internally by the script. By default, the size of the stamp is auto-
matically set to be as small as possible (i.e., two times the external radius of 
the ring specified by `--normradii`) to make the computation fast. As a conse-
quence, this option is only relevant for checking and testing that everything is 
fine (debugging; it will usually not be necessary).

- `S STR` 
  `--segment=STR` 
  Optional filename of a segmentation image from Segment’s output (must con-
tain the CLUMPS and OBJECT HDUs). For more on the definition of “objects”
and “clumps”, see Section 7.3 [Segment], page 381. If given, Segment’s output
is used to mask all background sources from the large foreground object (a
bright star):

- Objects that are not the central object.
- Clumps (within the central object) that are not the central clump.

The result is that all objects and clumps that contaminate the central source
are masked, while the diffuse flux of the central object remains. The size of
the region that is used for identifying the central clump/object label can be
specified by the option --corewidth, see below.

-w INT
--corewidth=INT
Width of the region that is used to compute the central clump/object label over
the segmentation image --segment, with the goal of not masking them. It is
rarely necessary to change this option.

If a segmentation image is provided, then it is necessary to not mask the central
object+clump (see description of --segment). To do that, the clump and object
images are cropped around the coordinate given to --center, but on the two
labeled images. Assuming a single value exists within the crop, the IDs of the
respective object and clump are found. Therefore it is necessary for this option
to be small (the program will abort if more than one label exists within it).

-s FLT,FLT
--sigmaclip=FLT,FLT
Sigma clipping parameters used in the end to find the final scale factor from
the distribution of all pixels used. For more on sigma-clipping, see Section 7.1.3
[Sigma clipping], page 343.

-t
--tmpdir
Directory to keep temporary files during the execution of the script. If the
directory doesn’t exist at runtime, this script will create it. By default, upon
completion of the script, this directory will be deleted. However, if you would
like to keep the intermediate files, you can use the --keeptmp option.

-k
--keeptmp
Do not remove the temporary directory (see description of --keeptmp). This
option is useful for debugging and checking the outputs of internal steps.

## 10.5.6 Invoking astscript-psf-subtract

This installed script will put the provided PSF into a given position within the input image
(implementing sub-pixel adjustments where necessary), and then it will subtract it. It is
aimed at modeling and subtracting the scattered light field of an input image. It is also
possible to obtain the modeled star with the PSF (and not make the subtraction of it from
the original image).

A complete tutorial is available to show the operation of this script as a modular com-
ponent to extract the PSF of a dataset: Section 2.4 [Building the extended PSF], page 97.
The executable name is astscript-psf-subtract, with the following general template:
$ astscript-psf-subtract [OPTION...] FITS-file

Examples:

## Multiply the PSF (psf.fits) by 3 and subtract it from the
## input image (image.fits) at the pixel position (x,y)=(53,69).
$ astscript-psf-subtract image.fits \
   --psf=psf.fits \
   --mode=img \
   --scale=3 \ 
   --center=53,69 \ 
   --output=star-53_69.fits

## Iterate over a catalog with positions of stars that are
## in the input image. Use WCS coordinates.
$ asttable catalog.fits | while read -r ra dec mag; do
   scale=$(cat scale-"$ra"_"$dec".txt)
   astscript-psf-subtract image.fits \
      --mode=wcs \
      --psf=psf.fits \
      --scale=$scale \
      --center=$ra,$dec; done

The input is an image from which the star is considered. The result is the same image
but with the star subtracted (modeled by the PSF). The modeling of the star is done with
the PSF image specified with the option --psf, and flux-scaled with the option --scale at
the position defined by --center. Instead of obtaining the PSF-subtracted image, it is also
possible to obtain the modeled star by the PSF. To do that, use the option --modelonly.
With this option, the output will be an image with the same size as the original one with
the PSF situated in the star coordinates and flux-scaled. In this case, the region not covered
by the PSF are set to zero values.

Note that this script works over individual objects. As a consequence, to generate a
scattered light field of many stars, it is necessary to make multiple calls. A full description
of each option is given below.

- **h STR**
  --hdu=STR
  The HDU/extension of the input image to use.

- **p STR**
  --psf=STR
  Filename of the PSF image. The PSF is assumed to be centered in this image.

- **P STR**
  --psfhdu=STR
  The HDU/extension of the PSF image.

- **O STR**
  --mode=STR
  Interpret the center position of the object (values given to --center) in image
  or WCS coordinates. This option thus accepts only two values: img or wcs.
-c FLT,FLT
--center=FLT,FLT
The central position of the object. This parameter is used in Section 6.1 [Crop], page 259, to center and crop the image. The positions along each dimension must be separated by a comma (,). The number of values given to this option must be the same as the dimensions of the input dataset. The units of the coordinates are read based on the value to the --mode option, see above.
If the central position doesn’t fall in the center of a pixel in the input image, the PSF is re-sampled with sub-pixel change in the pixel grid before subtraction.

-s FLT
--scale=FLT
Factor by which the PSF (--psf) is multiplied. This factor is necessary to put the PSF with the desired flux level. A convenient way of obtaining this value is by using the script astscript-scale-factor, see Section 10.5.5 [Invoking astscript-psf-scale-factor], page 508. For a full tutorial on using the astscript-psf-* scripts together, see Section 2.4 [Building the extended PSF], page 97.

-t
--tmpdir Directory to keep temporary files during the execution of the script. If the directory doesn’t exist at run-time, this script will create it. By default, upon completion of the script, this directory will be deleted. However, if you would like to keep the intermediate files, you can use the --keeptmp option.

-k
--keeptmp
Do not remove the temporary directory (see description of --keeptmp). This option is useful for debugging and checking the outputs of internal steps.

-m
--modelonly
Don’t make the subtraction of the modeled star by the PSF. This option is useful when the user wants to obtain the scattered light field given by the PSF modeled star.
11 Library

Each program in Gnuastro that was discussed in the prior chapters (or any program in general) is a collection of functions that is compiled into one executable file which can communicate directly with the outside world. The outside world in this context is the operating system. By communication, we mean that control is directly passed to a program from the operating system with a (possible) set of inputs and after it is finished, the program will pass control back to the operating system. For programs written in C and C++, the unique main function is in charge of this communication.

Similar to a program, a library is also a collection of functions that is compiled into one executable file. However, unlike programs, libraries don’t have a main function. Therefore they can’t communicate directly with the outside world. This gives you the chance to write your own main function and call library functions from within it. After compiling your program into a binary executable, you just have to link it to the library and you are ready to run (execute) your program. In this way, you can use Gnuastro at a much lower-level, and in combination with other libraries on your system, you can significantly boost your creativity.

This chapter starts with a basic introduction to libraries and how you can use them in Section 11.1 [Review of library fundamentals], page 513. The separate functions in the Gnuastro library are then introduced (classified by context) in Section 11.3 [Gnuastro library], page 525. If you end up routinely using a fixed set of library functions, with a well-defined input and output, it will be much more beneficial if you define a program for the job. Therefore, in its Section 3.2.2 [Version controlled source], page 132, Gnuastro comes with the Section 12.4.2 [The TEMPLATE program], page 680, to easily define your own programs(s).

11.1 Review of library fundamentals

Gnuastro’s libraries are written in the C programming language. In Section 12.1 [Why C programming language?], page 670, we have thoroughly discussed the reasons behind this choice. C was actually created to write Unix, thus understanding the way C works can greatly help in effectively using programs and libraries in all Unix-like operating systems. Therefore, in the following subsections some important aspects of C, as it relates to libraries (and thus programs that depend on them) on Unix are reviewed. First we will discuss header files in Section 11.1.1 [Headers], page 514, and then go onto Section 11.1.2 [Linking], page 517. This section finishes with Section 11.1.3 [Summary and example on libraries], page 520. If you are already familiar with these concepts, please skip this section and go directly to Section 11.3 [Gnuastro library], page 525.

In theory, a full operating system (or any software) can be written as one function. Such a software would not need any headers or linking (that are discussed in the subsections below). However, writing that single function and maintaining it (adding new features, fixing bugs, documentation, etc) would be a programmer or scientist’s worst nightmare! Furthermore, all the hard work that went into creating it cannot be reused in other software: every other programmer or scientist would have to re-invent the wheel. The ultimate purpose behind libraries (which come with headers and have to be linked) is to address this problem and increase modularity: “the degree to which a system’s components may be separated and
recombined” (from Wikipedia). The more modular the source code of a program or library, the easier maintaining it will be, and all the hard work that went into creating it can be reused for a wider range of problems.

11.1.1 Headers

C source code is read from top to bottom in the source file, therefore program components (for example variables, data structures and functions) should all be defined or declared closer to the top of the source file: before they are used. Defining something in C or C++ is jargon for providing its full details. Declaring it, on the other-hand, is jargon for only providing the minimum information needed for the compiler to pass it temporarily and fill in the detailed definition later.

For a function, the declaration only contains the inputs and their data-types along with the output’s type\(^1\). The definition adds to the declaration by including the exact details of what operations are done to the inputs to generate the output. As an example, take this simple summation function:

```c
double sum(double a, double b)
{
    return a + b;
}
```

What you see above is the definition of this function: it shows you (and the compiler) exactly what it does to the two double type inputs and that the output also has a double type. Note that a function’s internal operations are rarely so simple and short, it can be arbitrarily long and complicated. This unreasonably short and simple function was chosen here for ease of reading. The declaration for this function is:

```c
double sum(double a, double b);
```

You can think of a function’s declaration as a building’s address in the city, and the definition as the building’s complete blueprints. When the compiler confronts a call to a function during its processing, it doesn’t need to know anything about how the inputs are processed to generate the output. Just as the postman doesn’t need to know the inner structure of a building when delivering the mail. The declaration (address) is enough. Therefore by declaring the functions once at the start of the source files, we don’t have to worry about defining them after they are used.

Even for a simple real-world operation (not a simple summation like above!), you will soon need many functions (for example, some for reading/preparing the inputs, some for the processing, and some for preparing the output). Although it is technically possible, managing all the necessary functions in one file is not easy and is contrary to the modularity principle (see Section 11.1 [Review of library fundamentals], page 513), for example the functions for preparing the input can be usable in your other projects with a different processing. Therefore, as we will see later (in Section 11.1.2 [Linking], page 517), the functions don’t necessarily need to be defined in the source file where they are used. As long as their definitions are ultimately linked to the final executable, everything will be fine. For now, it is just important to remember that the functions that are called within

\(^1\) Recall that in C, functions only have one output.
one source file must be declared within the source file (declarations are mandatory), but not necessarily defined there.

In the spirit of modularity, it is common to define contextually similar functions in one source file. For example, in Gnuastro, functions that calculate the median, mean and other statistical functions are defined in \texttt{lib/statistics.c}, while functions that deal directly with FITS files are defined in \texttt{lib/fits.c}.

Keeping the definition of similar functions in a separate file greatly helps their management and modularity, but this fact alone doesn’t make things much easier for the caller’s source code: recall that while definitions are optional, declarations are mandatory. So if this was all, the caller would have to manually copy and paste (\texttt{include}) all the declarations from the various source files into the file they are working on now. To address this problem, programmers have adopted the header file convention: the header file of a source code contains all the declarations that a caller would need to be able to use any of its functions. For example, in Gnuastro, \texttt{lib/statistics.c} (file containing function definitions) comes with \texttt{lib/gnuastro/statistics.h} (only containing function declarations).

The discussion above was mainly focused on functions, however, there are many more programming constructs such as pre-processor macros and data structures. Like functions, they also need to be known to the compiler when it confronts a call to them. So the header file also contains their definitions or declarations when they are necessary for the functions.

Pre-processor macros (or macros for short) are replaced with their defined value by the pre-processor before compilation. Conventionally they are written only in capital letters to be easily recognized. It is just important to understand that the compiler doesn’t see the macros, it sees their fixed values. So when a header specifies macros you can do your programming without worrying about the actual values. The standard C types (for example \texttt{int}, or \texttt{float}) are very low-level and basic. We can collect multiple C types into a \texttt{structure} for a higher-level way to keep and pass-along data. See Section 11.3.6.1 [Generic data container (\texttt{gal_data_t})], page 542, for some examples of macros and data structures.

The contents in the header need to be \texttt{included} into the caller’s source code with a special pre-processor command: \texttt{#include <path/to/header.h>}. As the name suggests, the \texttt{pre-processor} goes through the source code prior to the processor (or compiler). One of its jobs is to include, or merge, the contents of files that are mentioned with this directive in the source code. Therefore the compiler sees a single entity containing the contents of the main file and all the included files. This allows you to include many (sometimes thousands of) declarations into your code with only one line. Since the headers are also installed with the library into your system, you don’t even need to keep a copy of them for each separate program, making things even more convenient.

Try opening some of the .c files in Gnuastro’s \texttt{lib/} directory with a text editor to check out the include directives at the start of the file (after the copyright notice). Let’s take \texttt{lib/fits.c} as an example. You will notice that Gnuastro’s header files (like \texttt{gnuastro/fits.h}) are indeed within this directory (the \texttt{fits.h} file is in the \texttt{gnuastro/} directory). You will notice that files like \texttt{stdio.h}, or \texttt{string.h} are not in this directory (or anywhere within Gnuastro).
On most systems the basic C header files (like `stdio.h` and `string.h` mentioned above) are located in `/usr/include/`\(^2\). Your compiler is configured to automatically search that directory (and possibly others), so you don’t have to explicitly mention these directories. Go ahead, look into the `/usr/include` directory and find `stdio.h` for example. When the necessary header files are not in those specific libraries, the pre-processor can also search in places other than the current directory. You can specify those directories with this pre-processor option\(^3\):

\[-I \text{DIR} \]

“Add the directory \text{DIR} to the list of directories to be searched for header files. Directories named by ‘-I’ are searched before the standard system include directories. If the directory \text{DIR} is a standard system include directory, the option is ignored to ensure that the default search order for system directories and the special treatment of system headers are not defeated...” (quoted from the GNU Compiler Collection manual). Note that the space between -I and the directory is optional and commonly not used.

If the pre-processor can’t find the included files, it will abort with an error. In fact a common error when building programs that depend on a library is that the compiler doesn’t not know where a library’s header is (see Section 3.3.5 [Known issues], page 149). So you have to manually tell the compiler where to look for the library’s headers with the -I option. For a small software with one or two source files, this can be done manually (see Section 11.1.3 [Summary and example on libraries], page 520). However, to enhance modularity, Gnuastro (and most other bin/libraries) contain many source files, so the compiler is invoked many times\(^4\). This makes manual addition or modification of this option practically impossible.

To solve this problem, in the GNU build system, there are conventional environment variables for the various kinds of compiler options (or flags). These environment variables are used in every call to the compiler (they can be empty). The environment variable used for the C Pre-Processor (or CPP) is `CPPFLAGS`. By giving `CPPFLAGS` a value once, you can be sure that each call to the compiler will be affected. See Section 3.3.5 [Known issues], page 149, for an example of how to set this variable at configure time.

As described in Section 3.3.1.2 [Installation directory], page 139, you can select the top installation directory of a software using the GNU build system, when you `./configure` it. All the separate components will be put in their separate sub-directory under that, for example the programs, compiled libraries and library headers will go into `$prefix/bin` (replace `$prefix` with a directory), `$prefix/lib`, and `$prefix/include` respectively. For enhanced modularity, libraries that contain diverse collections of functions (like GSL, WCSVLIB, and Gnuastro), put their header files in a sub-directory unique to themselves. For example all Gnuastro’s header files are installed in `$prefix/include/gnuastro`. In your source code, you need to keep the library’s sub-directory when including the headers from such libraries, for example `#include <gnuastro/fits.h>`\(^5\). Not all libraries need to follow

\(^2\) The `include/` directory name is taken from the pre-processor’s `#include` directive, which is also the motivation behind the ‘-I’ in the -I option to the pre-processor.

\(^3\) Try running Gnuastro’s `make` and find the directories given to the compiler with the -I option.

\(^4\) Nearly every command you see being executed after running `make` is one call to the compiler.

\(^5\) the top `$prefix/include` directory is usually known to the compiler.
this convention, for example CFITSIO only has one header (fitsio.h) which is directly installed in $prefix/include.

11.1.2 Linking

To enhance modularity, similar functions are defined in one source file (with a .c suffix, see Section 11.1.1 [Headers], page 514, for more). After running make, each human-readable, .c file is translated (or compiled) into a computer-readable “object” file (ending with .o). Note that object files are also created when building programs, they aren’t particular to libraries. Try opening Gnuastro’s lib/ and bin/progname/ directories after running make to see these object files. Afterwards, the object files are linked together to create an executable program or a library.

The object files contain the full definition of the functions in the respective .c file along with a list of any other function (or generally “symbol”) that is referenced there. To get a list of those functions you can use the nm program which is part of GNU Binutils. For example from the top Gnuastro directory, run:

```bash
$ nm bin/arithmetic/arithmetic.o
```

This will print a list of all the functions (more generally, ‘symbols’) that were called within bin/arithmetic/arithmetic.c along with some further information (for example a T in the second column shows that this function is actually defined here, U says that it is undefined here). Try opening the .c file to check some of these functions for your self. Run info nm for more information.

To recap, the compiler created the separate object files mentioned above for each .c file. The linker will then combine all the symbols of the various object files (and libraries) into one program or library. In the case of Arithmetic (a program) the contents of the object files in bin/arithmetic/ are copied (and re-ordered) into one final executable file which we can run from the operating system.

There are two ways to link all the necessary symbols: static and dynamic/shared. When the symbols (computer-readable function definitions in most cases) are copied into the output, it is called static linking. When the symbols are kept in their original file and only a reference to them is kept in the executable, it is called dynamic, or shared linking.

Let’s have a closer look at the executable to understand this better: we’ll assume you have built Gnuastro without any customization and installed Gnuastro into the default /usr/local/ directory (see Section 3.3.1.2 [Installation directory], page 139). If you tried the nm command on one of Arithmetic’s object files above, then with the command below you can confirm that all the functions that were defined in the object file above (had a T in the second column) are also defined in the astarithmetic executable:

```bash
$ nm /usr/local/bin/astarithmetic
```

These symbols/function have been statically linked (copied) in the final executable. But you will notice that there are still many undefined symbols in the executable (those with a U in the second column). One class of such functions are Gnuastro’s own library functions that start with ‘gal_’:

```bash
$ nm /usr/local/bin/astarithmetic | grep gal_
```

---

6 Gnuastro uses GNU Libtool for portable library creation. Libtool will also make a .lo file for each .c file when building libraries (.lo files are human-readable).
These undefined symbols (functions) are present in another file and will be linked to the Arithmetic program every time you run it. Therefore they are known as dynamically linked libraries\(^7\). As we saw above, static linking is done when the executable is being built. However, when a program is dynamically linked to a library, at build-time, the library's symbols are only checked with the available libraries: they are not actually copied into the program's executable. Every time you run the program, the (dynamic) linker will be activated and will try to link the program to the installed library before the program starts.

If you want all the libraries to be statically linked to the executables, you have to tell Libtool (which Gnuastro uses for the linking) to disable shared libraries at configure time\(^8\):

```bash
$ configure --disable-shared
```

Try configuring Gnuastro with the command above, then build and install it (as described in Section 1.1 [Quick start], page 1). Afterwards, check the `gal_` symbols in the installed Arithmetic executable like before. You will see that they are actually copied this time (have a T in the second column). If the second column doesn't convince you, look at the executable file size with the following command:

```bash
$ ls -lh /usr/local/bin/astarithmetic
```

It should be around 4.2 Megabytes with this static linking. If you configure and build Gnuastro again with shared libraries enabled (which is the default), you will notice that it is roughly 100 Kilobytes!

This huge difference would have been very significant in the old days, but with the roughly Terabyte storage drives commonly in use today, it is negligible. Fortunately, output file size is not the only benefit of dynamic linking: since it links to the libraries at run-time (rather than build-time), you don’t have to re-build a higher-level program or library when an update comes for one of the lower-level libraries it depends on. You just install the new low-level library and it will automatically be used/linked next time in the programs that use it. To be fair, this also creates a few complications\(^9\):

- Reproducibility: Even though your high-level tool has the same version as before, with the updated library, you might not get the same results.
- Broken links: if some functions have been changed or removed in the updated library, then the linker will abort with an error at run-time. Therefore you need to re-build your higher-level program or library.

To see a list of all the shared libraries that are needed for a program or a shared library to run, you can use GNU C library's `ldd\(^{10}\)` program, for example:

```bash
$ ldd /usr/local/bin/astarithmetic
```

Library file names (in their installation directory) start with a `lib` and their ending (suffix) shows if they are static (`.a`) or dynamic (`.so`), as described below. The name of

---

\(^7\) Do not confuse dynamically linked libraries with dynamically loaded libraries. The former (that is discussed here) are only loaded once at the program startup. However, the latter can be loaded anytime during the program's execution, they are also known as plugins.

\(^8\) Libtool is very common and is commonly used. Therefore, you can use this option to configure on most programs using the GNU build system if you want static linking.

\(^9\) Both of these can be avoided by joining the mailing lists of the lower-level libraries and checking the changes in newer versions before installing them. Updates that result in such behaviors are generally heavily emphasized in the release notes.

\(^{10}\) If your operating system is not using the GNU C library, you might need another tool.
Chapter 11: Library

519

the library is in the middle of these two, for example libgsl.a or libgnuastro.a (GSL and Gnuastro’s static libraries), and libgsl.so.23.0.0 or libgnuastro.so.4.0.0 (GSL and Gnuastro’s shared library, the numbers may be different).

- A static library is known as an archive file and has the .a suffix. A static library is not an executable file.
- A shared library ends with the .so.X.Y.Z suffix and is executable. The three numbers in the suffix, describe the version of the shared library. Shared library versions are defined to allow multiple versions of a shared library simultaneously on a system and to help detect possible updates in the library and programs that depend on it by the linker.

It is very important to mention that this version number is different from the software version number (see Section 1.6 [Version numbering], page 7), so do not confuse the two. See the “Library interface versions” chapter of GNU Libtool for more.

For each shared library, we also have two symbolic links ending with .so.X and .so. They are automatically set by the installer, but you can change them (point them to another version of the library) when you have multiple versions of a library on your system.

Libraries that are built with GNU Libtool (including Gnuastro and its dependencies), build both static and dynamic libraries by default and install them in prefix/lib/ directory (for more on prefix, see Section 3.3.1.2 [Installation directory], page 139). In this way, programs depending on the libraries can link with them however they prefer. See the contents of /usr/local/lib with the command below to see both the static and shared libraries available there, along with their executable nature and the symbolic links:

$ ls -l /usr/local/lib/

To link with a library, the linker needs to know where to find the library. At compilation time, these locations can be passed to the linker with two separate options (see Section 11.1.3 [Summary and example on libraries], page 520, for an example) as described below. You can see these options and their usage in practice while building Gnuastro (after running make):

-L DIR Will tell the linker to look into DIR for the libraries. For example -L/usr/local/lib, or -L/home/yourname/.local/lib. You can make multiple calls to this option, so the linker looks into several directories at compilation time. Note that the space between L and the directory is optional and commonly ignored (written as -LDIR).

-l LIBRARY Specify the unique library identifier/name (not containing directory or shared/dynamic nature) to be linked with the executable. As discussed above, library file names have fixed parts which must not be given to this option. So -lgsl will guide the linker to either look for libgsl.a or libgsl.so (depending on the type of linking it is suppose to do). You can link many libraries by repeated calls to this option.

Very important: The place of this option on the compiler’s command matters. This is often a source of confusion for beginners, so let’s assume you have asked the linker to link with library A using this option. As soon as the linker confronts
this option, it looks into the list of the undefined symbols it has found until that point and does a search in library A for any of those symbols. If any pending undefined symbol is found in library A, it is used. After the search in undefined symbols is complete, the contents of library A are completely discarded from the linker’s memory. Therefore, if a later object file or library uses an unlinked symbol in library A, the linker will abort after it has finished its search in all the input libraries or object files.

As an example, Gnuastro’s gal_fits_img_read function depends on the fits_read_pix function of CFITSIO (specified with -lcfitsio, which in turn depends on the cURL library, called with -lcurl). So the proper way to link something that uses this function is -lgnuastro -lcfitsio -lcurl. If instead, you give: -lcfitsio -lgnuastro the linker will complain and abort. To avoid such linking complexities when using Gnuastro’s library, we recommend using Section 11.2 [BuildProgram], page 521.

If you have compiled and linked your program with a dynamic library, then the dynamic linker also needs to know the location of the libraries after building the program: every time the program is run afterwards. Therefore, it may happen that you don’t get any errors when compiling/linking a program, but are unable to run your program because of a failure to find a library. This happens because the dynamic linker hasn’t found the dynamic library at run time.

To find the dynamic libraries at run-time, the linker looks into the paths, or directories, in the LD_LIBRARY_PATH environment variable. For a discussion on environment variables, especially search paths like LD_LIBRARY_PATH, and how you can add new directories to them, see Section 3.3.1.2 [Installation directory], page 139.

**11.1.3 Summary and example on libraries**

After the mostly abstract discussions of Section 11.1.1 [Headers], page 514, and Section 11.1.2 [Linking], page 517, we’ll give a small tutorial here. But before that, let’s recall the general steps of how your source code is prepared, compiled and linked to the libraries it depends on so you can run it:

1. The **pre-processor** includes the header (.h) files into the function definition (.c) files, expands pre-processor macros. Generally the pre-processor prepares the human-readable source for compilation (reviewed in Section 11.1.1 [Headers], page 514).

2. The **compiler** will translate (compile) the human-readable contents of each source (merged .c and the .h files, or generally the output of the pre-processor) into the computer-readable code of .o files.

3. The **linker** will link the called function definitions from various compiled files to create one unified object. When the unified product has a main function, this function is the product’s only entry point, enabling the operating system or user to directly interact with it, so the product is a program. When the product doesn’t have a main function, the linker’s product is a library and its exported functions can be linked to other executables (it has many entry points).

The GNU Compiler Collection (or GCC for short) will do all three steps. So as a first example, from Gnuastro’s source, go to tests/lib/. This directory contains the library
tests, you can use these as some simple tutorials. For this demonstration, we will compile and run the `arraymanip.c`. This small program will call Gnuastro library for some simple operations on an array (open it and have a look). To compile this program, run this command inside the directory containing it.

```
$ gcc arraymanip.c -lgnuastro -lm -o arraymanip
```

The two `-lgnuastro` and `-lm` options (in this order) tell GCC to first link with the Gnuastro library and then with C’s math library. The `-o` option is used to specify the name of the output executable, without it the output file name will be `a.out` (on most OSs), independent of your input file name(s).

If your top Gnuastro installation directory (let’s call it `$prefix`, see Section 3.3.1.2 [Installation directory], page 139) is not recognized by GCC, you will get pre-processor errors for unknown header files. Once you fix it, you will get linker errors for undefined functions. To fix both, you should run GCC as follows: additionally telling it which directories it can find Gnuastro’s headers and compiled library (see Section 11.1.1 [Headers], page 514, and Section 11.1.2 [Linking], page 517):

```
$ gcc -I$prefix/include -L$prefix/lib arraymanip.c -lgnuastro -lm \
-o arraymanip
```

This single command has done all the pre-processor, compilation and linker operations. Therefore no intermediate files (object files in particular) were created, only a single output executable was created. You are now ready to run the program with:

```
$ ./arraymanip
```

The Gnuastro functions called by this program only needed to be linked with the C math library. But if your program needs WCS coordinate transformations, needs to read a FITS file, needs special math operations (which include its linear algebra operations), or you want it to run on multiple CPU threads, you also need to add these libraries in the call to GCC: `-lgnuastro -lwcs -lcfitsio -lgs1 -lgs1cblas -pthread -lm`. In Section 11.3 [Gnuastro library], page 525, where each function is documented, it is mentioned which libraries (if any) must also be linked when you call a function. If you feel all these linkings can be confusing, please consider Gnuastro’s Section 11.2 [BuildProgram], page 521, program.

### 11.2 BuildProgram

The number and order of libraries that are necessary for linking a program with Gnuastro library might be too confusing when you need to compile a small program for one particular job (with one source file). BuildProgram will use the information gathered during configuring Gnuastro and link with all the appropriate libraries on your system. This will allow you to easily compile, link and run programs that use Gnuastro’s library with one simple command and not worry about which libraries to link to, or the linking order.

BuildProgram uses GNU Libtool to find the necessary libraries to link against (GNU Libtool is the same program that builds all of Gnuastro’s libraries and programs when you run `make`). So in the future, if Gnuastro’s prerequisite libraries change or other libraries are added, you don’t have to worry, you can just run BuildProgram and internal linking will be done correctly.
BuildProgram requires GNU Libtool: BuildProgram depends on GNU Libtool, other implementations don’t have some necessary features. If GNU Libtool isn’t available at Gnuastro’s configure time, you will get a notice at the end of the configuration step and BuildProgram will not be built or installed. Please see Section 3.1.2 [Optional dependencies], page 122, for more information.

11.2.1 Invoking BuildProgram

BuildProgram will compile and link a C source program with Gnuastro’s library and all its dependencies, greatly facilitating the compilation and running of small programs that use Gnuastro’s library. The executable name is `astbuildprog` with the following general template:

```bash
$ astbuildprog [OPTION...] C_SOURCE_FILE
```

One line examples:

```bash
## Compile, link and run ‘myprogram.c’:
$ astbuildprog myprogram.c

## Similar to previous, but with optimization and compiler warnings:
$ astbuildprog -Wall -O2 myprogram.c

## Compile and link ‘myprogram.c’, then run it with ‘image.fits’
## as its argument:
$ astbuildprog myprogram.c image.fits

## Also look in other directories for headers and linking:
$ astbuildprog -Lother -Iother/dir myprogram.c

## Just build (compile and link) ‘myprogram.c’, don’t run it:
$ astbuildprog --onlybuild myprogram.c
```

If BuildProgram is to run, it needs a C programming language source file as input. By default it will compile and link the given source into a final executable program and run it. The built executable name can be set with the optional `--output` option. When no output name is set, BuildProgram will use Gnuastro’s Section 4.9 [Automatic output], page 191, system to remove the suffix of the input source file (usually `.c`) and use the resulting name as the built program name.

For the full list of options that BuildProgram shares with other Gnuastro programs, see Section 4.1.2 [Common options], page 156. You may also use Gnuastro’s Section 4.2 [Configuration files], page 169, to specify other libraries/headers to use for special directories and not have to type them in every time.

The C compiler can be chosen with the `--cc` option, or environment variables, please see the description of `--cc` for more. The two common `LDFLAGS` and `CPPFLAGS` environment variables are also checked and used in the build by default. Note that they are placed after the values to the corresponding options `--includedir` and `--linkdir`. Therefore BuildProgram’s own options take precedence. Using environment variables can be disabled with the `--noenv` option. Just note that BuildProgram also keeps the important flags in these environment variables in its configuration file. Therefore, in many cases, even though you may needed them to build Gnuastro, you won’t need them in BuildProgram.
The first argument is considered to be the C source file that must be compiled and linked. Any other arguments (non-option tokens on the command-line) will be passed onto the program when BuildProgram wants to run it. Recall that by default BuildProgram will run the program after building it. This behavior can be disabled with the --onlybuild option.

When the --quiet option (see Section 4.1.2.3 [Operating mode options], page 162) is not called, BuildPrograms will print the compilation and running commands. Once your program grows and you break it up into multiple files (which are much more easily managed with Make), you can use the linking flags of the non-quiet output in your Makefile.

-c STR
--cc=STR  
C compiler to use for the compilation, if not given environment variables will be used as described in the next paragraph. If the compiler is in your system’s search path, you can simply give its name, for example --cc=gcc. If its not in your system’s search path, you can give its full path, for example --cc=/path/to/your/custom/cc.

If this option has no value after parsing the command-line and all configuration files (see Section 4.2.2 [Configuration file precedence], page 170), then BuildProgram will look into the following environment variables in the given order CC and GCC. If they are also not defined, BuildProgram will ultimately default to the gcc command which is present in many systems (sometimes as a link to other compilers).

-I STR
--includedir=STR
Directory to search for files that you #include in your C program. Note that headers relating to Gnuastro and its dependencies don’t need this option. This is only necessary if you want to use other headers. It may be called multiple times and order matters. This directory will be searched before those of Gnuastro’s build and also the system search directories. See Section 11.1.1 [Headers], page 514, for a thorough introduction.

From the GNU C Pre-Processor manual: “Add the directory STR to the list of directories to be searched for header files. Directories named by -I are searched before the standard system include directories. If the directory STR is a standard system include directory, the option is ignored to ensure that the default search order for system directories and the special treatment of system headers are not defeated”.

-L STR
--linkdir=STR
Directory to search for compiled libraries to link the program with. Note that all the directories that Gnuastro was built with will already be used by BuildProgram (GNU Libtool). This option is only necessary if your libraries are in other directories. Multiple calls to this option are possible and order matters. This directory will be searched before those of Gnuastro’s build and also the system search directories. See Section 11.1.2 [Linking], page 517, for a thorough introduction.
-l STR
--linklib=STR
Library to link with your program. Note that all the libraries that Gnuastro was built with will already be linked by BuildProgram (GNU Libtool). This option is only necessary if you want to link with other directories. Multiple calls to this option are possible and order matters. This library will be linked before Gnuastro’s library or its dependencies. See Section 11.1.2 [Linking], page 517, for a thorough introduction.

-O INT/STR
--optimize=INT/STR
Compiler optimization level: 0 (for no optimization, good debugging), 1, 2, 3 (for the highest level of optimizations). From the GNU Compiler Collection (GCC) manual: “Without any optimization option, the compiler’s goal is to reduce the cost of compilation and to make debugging produce the expected results. Statements are independent: if you stop the program with a break point between statements, you can then assign a new value to any variable or change the program counter to any other statement in the function and get exactly the results you expect from the source code. Turning on optimization flags makes the compiler attempt to improve the performance and/or code size at the expense of compilation time and possibly the ability to debug the program.” Please see your compiler’s manual for the full list of acceptable values to this option.

-g
--debug
Emit extra information in the compiled binary for use by a debugger. When calling this option, it is best to explicitly disable optimization with -O0. To combine both options you can run -gO0 (see Section 4.1.1.2 [Options], page 154, for how short options can be merged into one).

-W STR
--warning=STR
Print compiler warnings on command-line during compilation. “Warnings are diagnostic messages that report constructions that are not inherently erroneous but that are risky or suggest there may have been an error.” (from the GCC manual). It is always recommended to compile your programs with warnings enabled.

All compiler warning options that start with W are usable by this option in BuildProgram also, see your compiler’s manual for the full list. Some of the most common values to this option are: pedantic (Warnings related to standard C) and all (all issues the compiler confronts).

-t
--tag=STR
The language configuration information. Libtool can build objects and libraries in many languages. In many cases, it can identify the language automatically, but when it doesn’t you can use this option to explicitly notify Libtool of the language. The acceptable values are: CC for C, CXX for C++, GCJ for Java, F77
Chapter 11: Library

for Fortran 77, FC for Fortran, GO for Go and RC for Windows Resource. Note that the Gnuastro library is not yet fully compatible with all these languages.

-b
--onlybuild
Only build the program, don’t run it. By default, the built program is immediately run afterwards.

-d
--deletecompiled
Delete the compiled binary file after running it. This option is only relevant when the compiled program is run after being built. In other words, it is only relevant when --onlybuild is not called. It can be useful when you are busy testing a program or just want a fast result and the actual binary/compiled file is not of later use.

-a STR
--la=STR
Use the given .la file (Libtool control file) instead of the one that was produced from Gnuastro’s configuration results. The Libtool control file keeps all the necessary information for building and linking a program with a library built by Libtool. The default prefix/lib/libgnuastro.la keeps all the information necessary to build a program using the Gnuastro library gathered during configure time (see Section 3.3.1.2 [Installation directory], page 139, for prefix). This option is useful when you prefer to use another Libtool control file.

-e
--noenv
Don’t use environment variables in the build, just use the values given to the options. As described above, environment variables like CC, GCC, LDFLAGS, CPPFLAGS will be read by default and used in the build if they have been defined.

11.3 Gnuastro library

Gnuastro library’s programming constructs (function declarations, macros, data structures, or global variables) are classified by context into multiple header files (see Section 11.1.1 [Headers], page 514). In this section, the functions in each header will be discussed under a separate sub-section, which includes the name of the header. Assuming a function declaration is in headername.h, you can include its declaration in your source code with:

```c
#include <gnuastro/headername.h>
```

The names of all constructs in headername.h are prefixed with gal_headername_ (or GAL_HEADERNAME_ for macros). The gal_ prefix stands for GNU Astronomy Library.

Gnuastro library functions are compiled into a single file which can be linked on the command-line with the -lgnuastro option. See Section 11.1.2 [Linking], page 517, and Section 11.1.3 [Summary and example on libraries], page 520, for an introduction on linking and some fully working examples of the libraries.

Gnuastro’s library is a high-level library which depends on lower level libraries for some operations (see Section 3.1 [Dependencies], page 119). Therefore if at least one of Gnuastro’s

---

11 Within Gnuastro’s source, all installed .h files in lib/gnuastro/ are accompanied by a .c file in /lib/.
functions in your program use functions from the dependencies, you will also need to link those dependencies after linking with Gnuastro. See Section 11.2 [BuildProgram], page 521, for a convenient way to deal with the dependencies. BuildProgram will take care of the libraries to link with your program (which uses the Gnuastro library), and can even run the built program afterwards. Therefore it allows you to conveniently focus on your exciting science/research when using Gnuastro’s libraries.

Libraries are still under heavy development: Gnuastro was initially created to be a collection of command-line programs. However, as the programs and their shared functions grew, internal (not installed) libraries were added. Since the 0.2 release, the libraries are install-able. Hence the libraries are currently under heavy development and will significantly evolve between releases and will become more mature and stable in due time. It will stabilize with the removal of this notice. Check the NEWS file for interface changes. If you use the Info version of this manual (see Section 4.3.4 [Info], page 174), you don’t have to worry: the documentation will correspond to your installed version.

11.3.1 Configuration information (config.h)

The gnuastro/config.h header contains information about the full Gnuastro installation on your system. Gnuastro developers should note that this is the only header that is not available within Gnuastro, it is only available to a Gnuastro library user after installation. Within Gnuastro, config.h (which is included in every Gnuastro .c file, see Section 12.3 [Coding conventions], page 673) has more than enough information about the overall Gnuastro installation.

GAL_CONFIG_VERSION [Macro]
This macro can be used as a string literal containing the version of Gnuastro that is being used. See Section 1.6 [Version numbering], page 7, for the version formats. For example:

    printf("Gnuastro version: %s\n", GAL_CONFIG_VERSION);

or

    char *gnuastro_version=GAL_CONFIG_VERSION;

GAL_CONFIG_HAVE_GSL_INTERP_STEFFEN [Macro]
GNU Scientific Library (GSL) is a mandatory dependency of Gnuastro (see Section 3.1.1.1 [GNU Scientific Library], page 120). The Steffen interpolation function that can be used in Gnuastro was introduced in GSL version 2.0 (released in October 2015). This macro will have a value of 1 if the host GSL contains this feature at configure time, and 0 otherwise.

GAL_CONFIG_HAVE_FITS_IS_REENTRANT [Macro]
This macro will have a value of 1 when the CFITSIO of the host system has the fits_is_reentrant function (available from CFITSIO version 3.30). This function is used to see if CFITSIO was configured to read a FITS file simultaneously on different threads.

12 https://en.wikipedia.org/wiki/String_literal
Chapter 11: Library

11.3.2 Multithreaded programming (threads.h)

In recent years, newer CPUs don’t have significantly higher frequencies any more. However, CPUs are being manufactured with more cores, enabling more than one operation (thread) at each instant. This can be very useful to speed up many aspects of processing and in particular image processing.

Most of the programs in Gnuastro utilize multi-threaded programming for the CPU intensive processing steps. This can potentially lead to a significant decrease in the running time of a program, see Section 4.4.1 [A note on threads], page 176. In terms of reading
the code, you don’t need to know anything about multi-threaded programming. You can simply follow the case where only one thread is to be used. In these cases, threads are not used and can be completely ignored.

When the C language was defined (the K&R’s book was written), using threads was not common, so C’s threading capabilities aren’t introduced there. Gnuastro uses POSIX threads for multi-threaded programming, defined in the pthread.h system wide header. There are various resources for learning to use POSIX threads. An excellent tutorial (https://computing.llnl.gov/tutorials/pthreads/) is provided by the Lawrence Livermore National Laboratory, with abundant figures to better understand the concepts, it is a very good start. The book ‘Advanced programming in the Unix environment’13, by Richard Stevens and Stephen Rago, Addison-Wesley, 2013 (Third edition) also has two chapters explaining the POSIX thread constructs which can be very helpful.

An alternative to POSIX threads was OpenMP, but POSIX threads are low level, allowing much more control, while being easier to understand, see Section 12.1 [Why C programming language?], page 670. All the situations where threads are used in Gnuastro currently are completely independent with no need of coordination between the threads. Such problems are known as “embarrassingly parallel” problems. They are some of the simplest problems to solve with threads and are also the ones that benefit most from them, see the LLNL introduction14.

One very useful POSIX thread concept is pthread_barrier. Unfortunately, it is only an optional feature in the POSIX standard, so some operating systems don’t include it. Therefore in Section 11.3.2.1 [Implementation of pthread_barrier], page 528, we introduce our own implementation. This is a rather technical section only necessary for more technical readers and you can safely ignore it. Following that, we describe the helper functions in this header that can greatly simplify writing a multi-threaded program, see Section 11.3.2.2 [Gnuastro’s thread related functions], page 529, for more.

11.3.2.1 Implementation of pthread_barrier

One optional feature of the POSIX Threads standard is the pthread_barrier concept. It is a very useful high-level construct that allows for independent threads to “wait” behind a “barrier” for the rest after they finish. Barriers can thus greatly simplify the code in a multi-threaded program, so they are heavily used in Gnuastro. However, since its an optional feature in the POSIX standard, some operating systems don’t include it. So to make Gnuastro portable, we have written our own implementation of those pthread_barrier functions.

At ./configure time, Gnuastro will check if pthread_barrier constructs are available on your system or not. If pthread_barrier is not available, our internal implementation will be compiled into the Gnuastro library and the definitions and declarations below will be usable in your code with #include <gnuastro/threads.h>.

**pthread_barrierattr_t**

[Type]

Type to specify the attributes of a POSIX threads barrier.

---

13 Don’t let the title scare you! The two chapters on Multi-threaded programming are very self-sufficient and don’t need any more knowledge than K&R.

14 https://computing.llnl.gov/tutorials/parallel_comp/
pthread_barrier_t
Structure defining the POSIX threads barrier.

int
pthread_barrier_init (pthread_barrier_t *b, pthread_barrierattr_t *attr,
unsigned int limit)
Initialize the barrier b, with the attributes attr and total limit (a number of) threads
that must wait behind it. This function must be called before spinning off threads.

int
pthread_barrier_wait (pthread_barrier_t *b)
This function is called within each thread, just before it is ready to return. Once a
thread’s function hits this, it will “wait” until all the other functions are also finished.

int
pthread_barrier_destroy (pthread_barrier_t *b)
Destroy all the information in the barrier structure. This should be called by the
function that spun-off the threads after all the threads have finished.

Destroy a barrier before re-using it: It is very important to destroy the barrier before
(possibly) reusing it. This destroy function not only destroys the internal structures,
it also waits (in 1 microsecond intervals, so you will not notice!) until all the threads
don’t need the barrier structure any more. If you immediately start spinning off new
threads with a not-destroyed barrier, then the internal structure of the remaining
threads will get mixed with the new ones and you will get very strange and apparently
random errors that are extremely hard to debug.

11.3.2.2 Gnuastro’s thread related functions
The POSIX Threads functions offered in the C library are very low-level and offer a great
range of control over the properties of the threads. So if you are interested in customiz-
ing your tools for complicated thread applications, it is strongly encouraged to get a nice
familiarity with them. Some resources were introduced in Section 11.3.2 [Multithreaded
programming (threads.h)], page 527.

However, in many cases used in astronomical data analysis, you don’t need communi-
cation between threads and each target operation can be done independently. Since such
operations are very common, Gnuastro provides the tools below to facilitate the creation
and management of jobs without any particular knowledge of POSIX Threads for such
operations. The most interesting high-level functions of this section are the gal_threads_
number and gal_threads_spin_off that identify the number of threads on the system and
spin-off threads. You can see a demonstration of using these functions in Section 11.4.3
[Library demo - multi-threaded operation], page 664.

gal_threads_params
Structure keeping the parameters of each thread. When each thread is created, a
pointer to this structure is passed to it. The params element can be the pointer to a
structure defined by the user which contains all the necessary parameters to pass onto
the worker function. The rest of the elements within this structure are set internally by `gal_threads_spin_off` and are relevant to the worker function.

```c
struct gal_threads_params {
    size_t id; /* Id of this thread. */
    void *params; /* User-identified pointer. */
    size_t *indexs; /* Target indices given to this thread. */
    pthread_barrier_t *b; /* Barrier for all threads. */
};
```

**size_t**

### `gal_threads_number ()`

Return the number of threads that the operating system has available for your program. This number is usually fixed for a single machine and doesn’t change. So this function is useful when you want to run your program on different machines (with different CPUs).

**void**

### `gal_threads_spin_off (void *((*worker)(void *)), void *caller_params, size_t numactions, size_t numthreads, size_t minmapsize, int quietmmap)`

Distribute `numactions` jobs between `numthreads` threads and spin-off each thread by calling the `worker` function. The `caller_params` pointer will also be passed to `worker` as part of the `gal_threads_params` structure. For a fully working example of this function, please see Section 11.4.3 [Library demo - multi-threaded operation], page 664.

If there are many jobs (millions or billions) to organize, memory issues may become important. With `minmapsize` you can specify the minimum byte-size to allocate the necessary space in a memory-mapped file or alternatively in RAM. If `quietmmap` is non-zero, then a warning will be printed upon creating a memory-mapped file. For more on Gnuastro’s memory management, see Section 4.6 [Memory management], page 180.

**void**

### `gal_threads_attr_barrier_init (pthread_attr_t *attr, pthread_barrier_t *b, size_t limit)`

This is a low-level function in case you don’t want to use `gal_threads_spin_off`. It will initialize the general thread attribute `attr` and the barrier `b` with `limit` threads to wait behind the barrier. For maximum efficiency, the threads initialized with this function will be detached. Therefore no communication is possible between these threads and in particular `pthread_join` won’t work on these threads. You have to use the barrier constructs to wait for all threads to finish.

**char **

### `gal_threads_dist_in_threads (size_t numactions, size_t numthreads, size_t minmapsize, int quietmmap, size_t **indexs, size_t *icols)`

This is a low-level function in case you don’t want to use `gal_threads_spin_off`. The job of this function is to distribute `numactions` jobs/actions in `numthreads` threads. To do this, it will assign each job an ID, ranging from 0 to `numactions`-1. The output
is the allocated *indexes array and the *icols number. In memory, its just a simple
1D array that has numthreads × *icols elements. But you can visualize it as a
2D array with numthreads rows and *icols columns. For more on the logic of the
distribution, see below.

When you have millions/billions of jobs to distribute, indexes will become very large.
For memory management (when to use a memory-mapped file, and when to use
RAM), you need to specify the minmapsize and quietmmap arguments. For more on
memory management, see Section 4.6 [Memory management], page 180. In general,
if your distributed jobs will not be on the scale of billions (and you want everything
to always be written in RAM), just set minmapsize=-1 and quietmmap=1.

When indexes is actually in a memory-mapped file, this function will return a string
containing the name of the file (that you can later give to gal_pointer_mmap_free
to free/delete). When indexes is in RAM, this function will return a NULL pointer.
So after you are finished with indexes, you can free it like this:

```c
char *mmapname;
int quietmmap=1;
size_t *indexes, thrdcols;
size_t numactions=5000, minmapsize=-1;
size_t numthreads=gal_threads_number();

/* Distribute the jobs. */
mmapname=gal_threads_dist_in_threads(numactions, numthreads,
   minmapsize, quietmmap,
   &indexes, &thrdcols);

/* Do any processing you want... */

/* Free the 'indexes' array. */
if(mmapname) gal_pointer_mmap_free(&mmapname, quietmmap);
else free(indexes);
```

Here is a brief description of the reasoning behind the indexes array and how the jobs
are distributed. Let’s assume you have A actions (where there is only one function
and the input values differ for each action) and T threads available to the system with
A > T (common values for these two would be A > 1000 and T < 10). Spinning off a
thread is not a cheap job and requires a significant number of CPU cycles. Therefore,
creating A threads is not the best way to address such a problem. The most efficient
way to manage the actions is such that only T threads are created, and each thread
works on a list of actions identified for it in series (one after the other). This way
your CPU will get all the actions done with minimal overhead.

The purpose of this function is to do what we explained above: each row in the
indexes array contains the indices of actions which must be done by one thread (so it
has numthreads rows with *icols columns). However, when using indexes, you don’t
have to know the number of columns. It is guaranteed that all the rows finish with
GAL_BLANK_SIZE_T (see Section 11.3.5 [Library blank values (blank.h)], page 538).
The GAL_BLANK_SIZE_T macro plays a role very similar to a string’s \0: every row
finishes with this macro, so can easily stop parsing the indexes in the row as soon
as you confront \texttt{GAL\_BLANK\_SIZE\_T}. For some real examples, please see the example program in \texttt{tests/lib/multithread.c} for a demonstration.

**11.3.3 Library data types (type.h)**

Data in astronomy can have many types, numeric (numbers) and strings (names, identifiers). The former can also be divided into integers and floats, see Section 4.5 [Numeric data types], page 178, for a thorough discussion of the different numeric data types and which one is useful for different contexts.

To deal with the very large diversity of types that are available (and used in different contexts), in Gnuastro each type is identified with global integer variable with a fixed name, this variable is then passed onto functions that can work on any type or is stored in Gnuastro’s Section 11.3.6.1 [Generic data container (\texttt{gal\_data\_t})], page 542, as one piece of meta-data.

The actual values within these integer constants is irrelevant and you should never rely on them. When you need to check, explicitly use the named variable in the table below. If you want to check with more than one type, you can use C’s \texttt{switch} statement.

Since Gnuastro heavily deals with file input-output, the types it defines are fixed width types, these types are portable to all systems and are defined in the standard C header \texttt{stdint.h}. You don’t need to include this header, it is included by any Gnuastro header that deals with the different types. However, the most commonly used types in a C (or C++) program (for example \texttt{int} or \texttt{long}) are not defined by their exact width (storage size), but by their minimum storage. So for example on some systems, \texttt{int} may be 2 bytes (16-bits, the minimum required by the standard) and on others it may be 4 bytes (32-bits, common in modern systems).

With every type, a unique “blank” value (or place-holder showing the absence of data) can be defined. Please see Section 11.3.5 [Library blank values (\texttt{blank.h})], page 538, for constants that Gnuastro recognizes as a blank value for each type. See Section 4.5 [Numeric data types], page 178, for more explanation on the limits and particular aspects of each type.

\begin{verbatim}
GAL\_TYPE\_INVALID \hfill [Global integer]
This is just a place-holder to specifically mark that no type has been set.

GAL\_TYPE\_BIT \hfill [Global integer]
Identifier for a bit-stream. Currently no program in Gnuastro works directly on bits, but features will be added in the future.

GAL\_TYPE\_UINT8 \hfill [Global integer]
Identifier for an unsigned, 8-bit integer type: \texttt{uint8\_t} (from \texttt{stdint.h}), or an \texttt{unsigned char} in most modern systems.

GAL\_TYPE\_INT8 \hfill [Global integer]
Identifier for a signed, 8-bit integer type: \texttt{int8\_t} (from \texttt{stdint.h}), or a \texttt{signed char} in most modern systems.

GAL\_TYPE\_UINT16 \hfill [Global integer]
Identifier for an unsigned, 16-bit integer type: \texttt{uint16\_t} (from \texttt{stdint.h}), or an \texttt{unsigned short} in most modern systems.
\end{verbatim}
GAL_TYPE_INT16
Identifier for a signed, 16-bit integer type: int16_t (from stdint.h), or a short in most modern systems.

GAL_TYPE_UINT32
Identifier for an unsigned, 32-bit integer type: uint32_t (from stdint.h), or an unsigned int in most modern systems.

GAL_TYPE_INT32
Identifier for a signed, 32-bit integer type: int32_t (from stdint.h), or an int in most modern systems.

GAL_TYPE_UINT64
Identifier for an unsigned, 64-bit integer type: uint64_t (from stdint.h), or an unsigned long in most modern 64-bit systems.

GAL_TYPE_INT64
Identifier for a signed, 64-bit integer type: int64_t (from stdint.h), or a long in most modern 64-bit systems.

GAL_TYPE_INT
Identifier for a int type. This is just an alias to int16, or int32 types, depending on the system.

GAL_TYPE_UINT
Identifier for a unsigned int type. This is just an alias to uint16, or uint32 types, depending on the system.

GAL_TYPEULONG
Identifier for a unsigned long type. This is just an alias to uint32, or uint64 types for 32-bit, or 64-bit systems respectively.

GAL_TYPE_LONG
Identifier for a long type. This is just an alias to int32, or int64 types for 32-bit, or 64-bit systems respectively.

GAL_TYPE_SIZE_T
Identifier for a size_t type. This is just an alias to uint32, or uint64 types for 32-bit, or 64-bit systems respectively.

GAL_TYPE_FLOAT32
Identifier for a 32-bit single precision floating point type or float in C.

GAL_TYPE_FLOAT64
Identifier for a 64-bit double precision floating point type or double in C.

GAL_TYPE_COMPLEX32
Identifier for a complex number composed of two float types. Note that the complex type is not yet fully implemented in all Gnuastro’s programs.
Chapter 11: Library

GAL_TYPE_COMPLEX64
Identifier for a complex number composed of two double types. Note that the complex type is not yet fully implemented in all Gnuastro’s programs.

GAL_TYPE_STRING
Identifier for a string of characters (char *).

GAL_TYPE_STRLL
Identifier for a linked list of string of characters (gal_list_str_t, see Section 11.3.8.1 [List of strings], page 555).

The functions below are defined to make working with the integer constants above easier. In the functions below, the constants above can be used for the type input argument.

size_t
gal_type_sizeof (uint8_t type)
Return the number of bytes occupied by type. Internally, this function uses C’s sizeof operator to measure the size of each type. For strings, this function will return the size of char *.

char *
gal_type_name (uint8_t type, int long_name)
Return a string literal that contains the name of type. It can return both short and long formats of the type names (for example f32 and float32). If long_name is non-zero, the long format will be returned, otherwise the short name will be returned. The output string is statically allocated, so it should not be freed. This function is the inverse of the gal_type_from_name function. For the full list of names/strings that this function will return, see Section 4.5 [Numeric data types], page 178.

uint8_t
gal_type_from_name (char *str)
Return the Gnuastro integer constant that corresponds to the string str. This function is the inverse of the gal_type_name function and accepts both the short and long formats of each type. For the full list of names/strings that this function will return, see Section 4.5 [Numeric data types], page 178.

void
gal_type_min (uint8_t type, void *in)
Put the minimum possible value of type in the space pointed to by in. Since the value can have any type, this function doesn’t return anything, it assumes the space for the given type is available to in and writes the value there. Here is one example

```c
int32_t min;
gal_type_min(GAL_TYPE_INT32, &min);
```

Note: Do not use the minimum value for a blank value of a general (initially unknown) type, please use the constants/functions provided in Section 11.3.5 [Library blank values (blank.h)], page 538, for the definition and usage of blank values.
void gal_type_max (uint8_t type, void *in)
    Put the maximum possible value of type in the space pointed to by in. Since the
    value can have any type, this function doesn’t return anything, it assumes the space
    for the given type is available to in and writes the value there. Here is one example
    
    uint16_t max;
    gal_type_max(GAL_TYPE_INT16, &max);
    
    Note: Do not use the maximum value for a blank value of a general (initially unknown)
    type, please use the constants/functions provided in Section 11.3.5 [Library blank
    values (blank.h)], page 538, for the definition and usage of blank values.

int gal_type_is_int (uint8_t type)
    Return 1 if the type is an integer (any width and any sign).

int gal_type_is_list (uint8_t type)
    Return 1 if the type is a linked list and zero otherwise.

int gal_type_out (int first_type, int second_type)
    Return the larger of the two given types which can be used for the type of the output
    of an operation involving the two input types.

char * gal_type_bit_string (void *in, size_t size)
    Return the bit-string in the size bytes that in points to. The string is dynamically
    allocated and must be freed afterwards. You can use it to inspect the bits within one
    region of memory. Here is one short example:
    
    int32_t a=2017;
    char *bitstr=gal_type_bit_string(&a, 4);
    printf("%d: %s (%X)\n", a, bitstr, a);
    free(bitstr);
    
    which will produce:
    
    2017: 11100001000001110000000000000000 (7E1)
    
    As the example above shows, the bit-string is not the most efficient way to inspect
    bits. If you are familiar with hexadecimal notation, it is much more compact, see
    https://en.wikipedia.org/wiki/Hexadecimal. You can use printf’s %x or %X to
    print integers in hexadecimal format.

char * gal_type_to_string (void *ptr, uint8_t type, int quote_if_str_has_space);
    Read the contents of the memory that ptr points to (assuming it has type type and
    print it into an allocated string which is returned.
    
    If the memory is a string of characters and quote_if_str_has_space is non-zero,
    the output string will have double-quotes around it if it contains space characters.
Also, note that in this case, \texttt{ptr} must be a pointer to an array of characters (or \texttt{char **}), as in the example below (which will put "sample string" into \texttt{out}):

\begin{verbatim}
    char *out, *string="sample string"
    out = gal_type_to_string(&string, GAL_TYPE_STRING, 1);
\end{verbatim}

\textbf{Function}

\begin{verbatim}
int gal_type_from_string (void **out, char *string, uint8_t type)
\end{verbatim}

Read a string as a given data type and put a pointer to it in \texttt{*out}. When \texttt{*out}!=\texttt{NULL}, then it is assumed to be already allocated and the value will be simply put there. If \texttt{*out}==\texttt{NULL}, then space will be allocated for the given type and the string will be read into that type.

Note that when we are dealing with a string type, \texttt{*out} should be interpreted as \texttt{char **} (one element in an array of pointers to different strings). In other words, \texttt{out} should be \texttt{char ***}.

This function can be used to fill in arrays of numbers from strings (in an already allocated data structure), or add nodes to a linked list (if the type is a list type). For an array, you have to pass the pointer to the \texttt{i}th element where you want the value to be stored, for example \texttt{&(array[i])}.

If the string was successfully parsed to the requested type, this function will return a \texttt{0} (zero), otherwise it will return \texttt{1} (one). This output format will help you check the status of the conversion in a code like the example below where we will try reading a string as a single precision floating point number.

\begin{verbatim}
    float out;
    void *outptr=&out;
    if( gal_type_from_string(&outptr, string, GAL_TYPE_FLOAT32) )
    {
        fprintf(stderr, "%s couldn't be read as float32\n", string);
        exit(EXIT_FAILURE);
    }
\end{verbatim}

When you need to read many numbers into an array, \texttt{out} would be an array, and you can simply increment \texttt{outptr=\texttt{out}+i} (where you increment \texttt{i}).

\textbf{Function}

\begin{verbatim}
void * gal_type_string_to_number (char *string, uint8_t *type)
\end{verbatim}

Read \texttt{string} into smallest type that can host the number, the allocated space for the number will be returned and the type of the number will be put into the memory that \texttt{type} points to. If \texttt{string} couldn’t be read as a number, this function will return \texttt{NULL}.

This function calls the C library’s \texttt{strtod} function to read \texttt{string} as a double-precision floating point number. When successful, it will check the value to put it in the smallest numerical data type that can handle it. However, if \texttt{string} is successfully parsed as a number \texttt{and} there is \texttt{.} in \texttt{string}, it will force the number into floating point types. For example "5" is read as an integer, while "5." or "5.0", or "5.00" will be read as a floating point (single-precision).

For the ranges acceptable by each type see Section 4.5 [Numeric data types], page 178. For integers, the range is clear, but for floating point types, this function will count
the number of significant digits and determine if the given string is single or double precision as described in that section.

### 11.3.4 Pointers (pointer.h)

Pointers play an important role in the C programming language. As the name suggests, they point to a byte in memory (like an address in a city). The C programming language gives you complete freedom in how to use the byte (and the bytes that follow it). Pointers are thus a very powerful feature of C. However, as the saying goes: “With great power comes great responsibility”, so they must be approached with care. The functions in this header are not very complex, they are just wrappers over some basic pointer functionality regarding pointer arithmetic and allocation (in memory or HDD/SSD).

```
void * gal_pointer_increment (void *pointer, size_t increment, uint8_t type)
```

Return a pointer to an element that is `increment` elements ahead of `pointer`, assuming each element has type of `type`. For the type codes, see Section 11.3.3 [Library data types (type.h)], page 532.

When working with the array elements of `gal_data_t`, we are actually dealing with void * pointers. However, pointer arithmetic doesn’t apply to void *, because the system doesn’t know how many bytes there are in each element to increment the pointer respectively. This function will use the given type to calculate where the incremented element is located in memory.

```
size_t gal_pointer_num_between (void *earlier, void *later, uint8_t type)
```

Return the number of elements (in the given type) between `earlier` and `later`. For the type codes, see Section 11.3.3 [Library data types (type.h)], page 532.

```
void * gal_pointer_allocate (uint8_t type, size_t size, int clear, const char *funcname, const char *varname)
```

Allocate an array of type `type` with `size` elements in RAM (for the type codes, see Section 11.3.3 [Library data types (type.h)], page 532). If `clear!=0`, then the allocated space is set to zero (cleared).

This is effectively just a wrapper around C’s malloc or calloc functions but takes Gnuastro’s integer type codes and will also abort with a clear error if there the allocation was not successful. The number of allocated bytes is the value given to size that is multiplied by the returned value of `gal_type_sizeof` for the given type. So if you want to allocate space for an array of strings you should pass the type GAL_TYPE_STRING. Otherwise, if you just want space for one string (for example 6 bytes for hello, including the string-termination character), you should set the type GAL_TYPE_UINT8.

When space cannot be allocated, this function will abort the program with a message containing the reason for the failure. `funcname` (name of the function calling this function) and `varname` (name of variable that needs this space) will be used in this error message if they are not NULL. In most modern compilers, you can use the generic `__func__` variable for `funcname`. In this way, you don’t have to manually copy and
paste the function name or worry about it changing later (__func__ was standardized in C99).

```c
void *
gal_pointer_allocate_ram_or_mmap (uint8_t type, size_t size, int clear,
    size_t minmmapsize, char **mmapname, int quietmmap, const char
    *funcname, const char *varname)
```

Allocate the given space either in RAM or in a memory-mapped file. This function is just a high-level wrapper to `gal_pointer_allocate` (to allocate in RAM) or `gal_pointer_mmap_allocate` (to use a memory-mapped file). For more on memory management in Gnuastro, please see Section 4.6 [Memory management], page 180. The various arguments are more fully explained in the two functions above.

```c
void *
gal_pointer_mmap_allocate (size_t size, uint8_t type, int clear, char
    **mmapname)
```

Allocate the necessary space to keep `size` elements of type `type` in HDD/SSD (a file, not in RAM). For the type codes, see Section 11.3.3 [Library data types (type.h)], page 532. If `clear!=0`, then the allocated space will also be cleared. The allocation is done using C’s `mmap` function. The name of the file containing the allocated space is an allocated string that will be put in `*mmapname`.

Note that the kernel doesn’t allow an infinite number of memory mappings to files. So it is not recommended to use this function with every allocation. The best case scenario to use this function is for large arrays that are very large and can fill up the RAM. Keep the smaller arrays in RAM, which is faster and can have a (theoretically) unlimited number of allocations.

When you are done with the dataset and don’t need it anymore, don’t use `free` (the dataset isn’t in RAM). Just delete the file (and the allocated space for the filename) with the commands below, or simply use `gal_pointer_mmap_free`.

```c
remove(mmapname);
free(mmapname);
```

```c
void
gal_pointer_mmap_free (char **mmapname, int quietmmap)
```

“Free” (actually delete) the memory-mapped file that is named `*mmapname`, then free the string. If `quietmmap` is non-zero, then a warning will be printed for the user to know that the given file has been deleted.

### 11.3.5 Library blank values (blank.h)

When the position of an element in a dataset is important (for example a pixel in an image), a place-holder is necessary for the element if we don’t have a value to fill it with (for example the CCD cannot read those pixels). We cannot simply shift all the other pixels to fill in the one we have no value for. In other cases, it often occurs that the field of sky that you are studying is not a clean rectangle to nicely fit into the boundaries of an image. You need a way to separate the pixels outside your scientific field from those inside it. Blank values act as these place holders in a dataset. They have no usable value but they have a position.
Every type needs a corresponding blank value (see Section 4.5 [Numeric data types], page 178, and Section 11.3.3 [Library data types (type.h)], page 532). Floating point types have a unique value identified by IEEE known as Not-a-Number (or NaN) which is a unique value that is recognized by the compiler. However, integer and string types don’t have any standard value. For integers, in Gnuastro we take an extremum of the given type: for signed types (that allow negatives), the minimum possible value is used as blank and for unsigned types (that only accept positives), the maximum possible value is used. To be generic and easy to read/write we define a macro for these blank values and strongly encourage you only use these, and never make any assumption on the value of a type’s blank value.

The IEEE NaN blank value type is defined to fail on any comparison, so if you are dealing with floating point types, you cannot use equality (a NaN will not be equal to a NaN). If you know your dataset is floating point, you can use the isnan function in C’s math.h header. For a description of numeric data types see Section 4.5 [Numeric data types], page 178. For the constants identifying integers, please see Section 11.3.3 [Library data types (type.h)], page 532.

GAL_BLANK_UINT8 [Global integer]
  Blank value for an unsigned, 8-bit integer.

GAL_BLANK_INT8 [Global integer]
  Blank value for a signed, 8-bit integer.

GAL_BLANK_UINT16 [Global integer]
  Blank value for an unsigned, 16-bit integer.

GAL_BLANK_INT16 [Global integer]
  Blank value for a signed, 16-bit integer.

GAL_BLANK_UINT32 [Global integer]
  Blank value for an unsigned, 32-bit integer.

GAL_BLANK_INT32 [Global integer]
  Blank value for a signed, 32-bit integer.

GAL_BLANK_UINT64 [Global integer]
  Blank value for an unsigned, 64-bit integer.

GAL_BLANK_INT64 [Global integer]
  Blank value for a signed, 64-bit integer.

GAL_BLANK_INT [Global integer]
  Blank value for int type (int16_t or int32_t depending on the system.

GAL_BLANK_UINT [Global integer]
  Blank value for int type (int16_t or int32_t depending on the system.

GAL_BLANK_LONG [Global integer]
  Blank value for long type (int32_t or int64_t in 32-bit or 64-bit systems).

GAL_BLANK_ULONG [Global integer]
  Blank value for unsigned long type (uint32_t or uint64_t in 32-bit or 64-bit systems).
GAL_BLANK_SIZE_T
[Global integer]
Blank value for size_t type (uint32_t or uint64_t in 32-bit or 64-bit systems).

GAL_BLANK_FLOAT32
[Global integer]
Blank value for a single precision, 32-bit floating point type (IEEE NaN value).

GAL_BLANK_FLOAT64
[Global integer]
Blank value for a double precision, 64-bit floating point type (IEEE NaN value).

GAL_BLANK_STRING
[Global integer]
Blank value for string types (this is itself a string, it isn’t the NULL pointer).

The functions below can be used to work with blank pixels.

void
[Function]
void *gal_blank_write (void *pointer, uint8_t type)
Write the blank value for the given type into the space that pointer points to. This can be used when the space is already allocated (for example one element in an array or a statically allocated variable).

void *
[Function]
void *gal_blank_alloc_write (uint8_t type)
Allocate the space required to keep the blank for the given data type type, write the blank value into it and return the pointer to it.

void
[Function]
void gal_blank_initialize (gal_data_t *input)
Initialize all the elements in the input dataset to the blank value that corresponds to its type. If input isn’t a string, and is a tile over a larger dataset, only the region that the tile covers will be set to blank. For strings, the full dataset will be initialized.

void
[Function]
void gal_blank_initialize_array (void *array, size_t size, uint8_t type)
Initialize all the elements in the array to the blank value that corresponds to its type (identified with type), assuming the array has size elements.

char *
[Function]
char *gal_blank_as_string (uint8_t type, int width)
Write the blank value for the given data type type into a string and return it. The space for the string is dynamically allocated so it must be freed after you are done with it. If width!=0, then the final string will be padded with white space characters to have the requested width if it is smaller.

int
[Function]
int gal_blank_is (void *pointer, uint8_t type)
Return 1 if the contents of pointer (assuming a type of type) is blank. Otherwise, return 0. Note that this function only works on one element of the given type. So if pointer is an array, only its first element will be checked. Therefore for strings, the type of pointer is assumed to be char *. To check if an array/dataset has blank elements or to find which elements in an array are blank, you can use gal_blank_present or gal_blank_flag respectively (described below).
int gal_blank_present (gal_data_t *input, int updateflag)
Return 1 if the dataset has a blank value and zero if it doesn’t. Before checking
the dataset, this function will look at input’s flags. If the GAL_DATA_FLAG_BLANK_CH
bit of input->flag is on, this function will not do any check and will just use the
information in the flags. This can greatly speed up processing when a dataset needs
to be checked multiple times.

When the dataset’s flags were not used and updateflags is non-zero, this function
will set the flags appropriately to avoid having to re-check the dataset in future calls.
When updateflags==0, this function has no side-effects on the dataset: it will not
toggle the flags.

If you want to re-check a dataset with the blank-value-check flag already set (for
example if you have made changes to it), then explicitly set the GAL_DATA_FLAG_-
BLANK_CH bit to zero before calling this function. When there are no other flags, you
can just set the flags to zero (input->flag=0), otherwise you can use this expression:

input->flag &= ~GAL_DATA_FLAG_BLANK_CH;

size_t gal_blank_number (gal_data_t *input, int updateflag)
Return the number of blank elements in input. If updateflag!=0, then the dataset
blank keyword flags will be updated. See the description of gal_blank_present
(above) for more on these flags. If input==NULL, then this function will return GAL_-
BLANK_SIZE_T.

gal_data_t * gal_blank_flag (gal_data_t *input)
Create a dataset of the same size as the input, but with an uint8_t type that has a
value of 1 for data that are blank and 0 for those that aren’t.

void gal_blank_flag_apply (gal_data_t *input, gal_data_t *flag)
Set all non-zero and non-blank elements of flag to blank in input. flag has to have
an unsigned 8-bit type and be the same size as input.

void gal_blank_flag_remove (gal_data_t *input, gal_data_t *flag)
Remove all elements within input that are flagged, convert it to a 1D dataset and ad-
just the size properly (the number of non-flagged elements). In practice this function
doesn’t realloc the input array (see gal_blank_remove_realloc for shrinking/re-
allocating also), it just shifts the blank elements to the end and adjusts the size
elements of the gal_data_t, see Section 11.3.6.1 [Generic data container (gal_data_-
t)], page 542.

Note that elements that are blank, but not flagged will not be removed. This function
will only remove flagged elements.

If all the elements were flagged, then input->size will be zero. This is thus a good
parameter to check after calling this function to see if there actually were any non-
flagged elements in the input or not and take the appropriate measure. This check is
highly recommended because it will avoid strange bugs in later steps.
void gal_blank_remove (gal_data_t *input)

Remove blank elements from a dataset, convert it to a 1D dataset, adjust the size properly (the number of non-blank elements), and toggle the blank-value-related bit-flags. In practice this function doesn’t realloc the input array (see gal_blank_remove_realloc for shrinking/re-allocating also), it just shifts the blank elements to the end and adjusts the size elements of the gal_data_t, see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542.

If all the elements were blank, then input->size will be zero. This is thus a good parameter to check after calling this function to see if there actually were any non-blank elements in the input or not and take the appropriate measure. This check is highly recommended because it will avoid strange bugs in later steps.

void gal_blank_remove_realloc (gal_data_t *input)

Similar to gal_blank_remove, but also shrinks/re-allocates the dataset’s allocated memory.

gal_data_t * gal_blank_remove_rows (gal_data_t *columns, gal_list_size_t *column_indices)

Remove any row that has at least one blank value in any of the input columns. The input columns is a list of gal_data_ts (see Section 11.3.8.9 [List of gal_data_t], page 566). After this function, all the elements in columns will still have the same size as each other, but if any of the searched columns has blank elements, all their sizes will decrease together.

If column_indices==NULL, then all the columns (nodes in the list) will be checked for blank elements, and any row that has at least one blank element will be removed. When column_indices!=NULL, only the columns whose index (counting from zero) is in column_indices will be used to check for blank values (see Section 11.3.8.3 [List of size_t], page 558. In any case (no matter which columns are checked for blanks), the selected rows from all columns will be removed.

11.3.6 Data container (data.h)

Astronomical datasets have various dimensions, for example 1D spectra or table columns, 2D images, or 3D Integral field data cubes. Datasets can also have various numeric data types, depending on the operation/purpose, for example processed images are commonly stored in floating point format, but their mask images are integers (allowing bit-wise flags to identify certain classes of pixels to keep or mask, see Section 4.5 [Numeric data types], page 178). Certain other information about a dataset are also commonly necessary, for example the units of the dataset, the name of the dataset and some comments. To deal with any generic dataset, Gnuastro defines the gal_data_t as input or output.

11.3.6.1 Generic data container (gal_data_t)

To be able to deal with any dataset (various dimensions, numeric data types, units and higher-level structures), Gnuastro defines the gal_data_t type which is the input/output container of choice for many of Gnuastro library’s functions. It is defined in gnuastro/data.h. If you will be using (# include’ing) those libraries, you don’t need
to include this header explicitly, it is already included by any library header that uses gal_data_t.

**gal_data_t**

[Type (C struct)]

The main container for datasets in Gnuastro. It can host data of any dimensions, with any numeric data type. It is actually a structure, but typedef’d as a new type to avoid having to write the **struct** before any declaration. The actual structure is shown below which is followed by a description of each element.

```c
typedef struct gal_data_t
{
  void *restrict array; /* Basic array information. */
  uint8_t type;
  size_t ndim;
  size_t *dsize;
  size_t size;
  int quietmmap;
  char *mmapname;
  size_t minmapsize;

  int nwcs; /* WCS information. */
  struct wcsprm *wcs;

  uint8_t flag; /* Content description. */
  int status;
  char *name;
  char *unit;
  char *comment;

  int disp_fmt; /* For text printing. */
  int disp_width;
  int disp_precision;

  struct gal_data_t *next; /* For higher-level datasets. */
  struct gal_data_t *block;
} gal_data_t;
```

The list below contains a description for each **gal_data_t** element.

**void *restrict array**

This is the pointer to the main array of the dataset containing the raw data (values). All the other elements in this data-structure are actually meta-data enabling us to use/understand the series of values in this array. It must allow data of any type (see Section 4.5 [Numeric data types], page 178), so it is defined as a **void * pointer. A **void * array is not directly usable in C, so you have to cast it to proper type before using it, please see Section 11.4.1 [Library demo - reading a FITS image], page 662, for a demonstration.

The **restrict** keyword was formally introduced in C99 and is used to tell the compiler that at any moment only this pointer will modify what it points to (a
This extra piece of information can greatly help in compiler optimizations and thus the running time of the program. But older compilers might not have this capability, so at ./configure time, Gnuastro checks this feature and if the user’s compiler doesn’t support restrict, it will be removed from this definition.

**uint8_t type**

A fixed code (integer) used to identify the type of data in array (see Section 4.5 [Numeric data types], page 178). For the list of acceptable values to this variable, please see Section 11.3.3 [Library data types (type.h)], page 532.

**size_t ndim**

The dataset’s number of dimensions.

**size_t *dsize**

The size of the dataset along each dimension. This is an array (with ndim elements), of positive integers in row-major order (based on C). When a data file is read into memory with Gnuastro’s libraries, this array is dynamically allocated based on the number of dimensions that the dataset has.

It is important to remember that C’s row-major ordering is the opposite of the FITS standard which is in column-major order: in the FITS standard the fastest dimension’s size is specified by NAXIS1, and slower dimensions follow. The FITS standard was defined mainly based on the FORTRAN language which is the opposite of C’s approach to multi-dimensional arrays (and also starts counting from 1 not 0). Hence if a FITS image has NAXIS1==20 and NAXIS2==50, the dsize array must be filled with dsize[0]==50 and dsize[1]==20.

The fastest dimension is the one that is contiguous in memory: to increment by one along that dimension, just go to the next element in the array. As we go to slower dimensions, the number of memory cells we have to skip for an increment along that dimension becomes larger.

**size_t size**

The total number of elements in the dataset. This is actually a multiplication of all the values in the dsize array, so it is not an independent parameter. However, low-level operations with the dataset (irrespective of its dimensions) commonly need this number, so this element is designed to avoid calculating it every time.

**int quietmmap**

When this value is zero, and the dataset must not be allocated in RAM (see mmapname and minmapsize), a warning will be printed to inform the user when the file is created and when it is deleted. The warning includes the filename, the size in bytes, and the fact that they can toggle this behavior through --minmapsize option in Gnuastro’s programs.

---

15 Also see https://en.wikipedia.org/wiki/Restrict.
16 Also see https://en.wikipedia.org/wiki/Row-_and_column-major_order.
char *mmapname

Name of file hosting the mmap'd contents of array. If the value of this variable is NULL, then the contents of array are actually stored in RAM, not in a file on the HDD/SSD. See the description of minmapsize below for more.

If a file is used, it will be kept in the gnuastro_mmap directory of the running directory. Its name is randomly selected to allow multiple arrays at the same time, see description of --minmapsize in Section 4.1.2.2 [Processing options], page 159. When gal_data_free is called the randomly named file will be deleted.

size_t minmapsize

The minimum size of an array (in bytes) to store the contents of array as a file (on the non-volatile HDD/SSD), not in RAM. This can be very useful for large datasets which can be very memory intensive and the user's RAM might not be sufficient to keep/process it. A random filename is assigned to the array which is available in the mmapname element of gal_data_t (above), see there for more. minmapsize is stored in each gal_data_t, so it can be passed on to subsequent/derived datasets.

See the description of the --minmapsize option in Section 4.1.2.2 [Processing options], page 159, for more on using this value.

nwcs

The number of WCS coordinate representations (for WCSLIB).

struct wcsprm *wcs

The main WCSLIB structure keeping all the relevant information necessary for WCSLIB to do its processing and convert data-set positions into real-world positions. When it is given a NULL value, all possible WCS calculations/measurements will be ignored.

uint8_t flag

Bit-wise flags to describe general properties of the dataset. The number of bytes available in this flag is stored in the GAL_DATA_FLAG_SIZE macro. Note that you should use bit-wise operators\(^\text{17}\) to check these flags. The currently recognized bits are stored in these macros:

GAL_DATA_FLAG_BLANK_CH

Marking that the dataset has been checked for blank values or not. When a dataset doesn’t have any blank values, the GAL_DATA_FLAG_HASBLANK bit will be zero. But upon initialization, all bits also get a value of zero. Therefore, a checker needs this flag to see if the value in GAL_DATA_FLAG_HASBLANK is reliable (dataset has actually been parsed for a blank value) or not.

Also, if it is necessary to re-check the presence of flags, you just have to set this flag to zero and call gal_blank_present for example to parse the dataset and check for blank values. Note that for improved efficiency, when this flag is set, gal_blank_present will not actually parse the dataset, it will just use GAL_DATA_FLAG_HASBLANK.

GAL_DATA_FLAG_HASBLANK
This bit has a value of 1 when the given dataset has blank values. If this bit is 0 and GAL_DATA_FLAG_BLANK_CH is 1, then the dataset has been checked and it didn't have any blank values, so there is no more need for further checks.

GAL_DATA_FLAG_SORT_CH
Marking that the dataset is already checked for being sorted or not and thus that the possible 0 values in GAL_DATA_FLAG_SORTED_I and GAL_DATA_FLAG_SORTED_D are meaningful. The logic behind this is similar to that in GAL_DATA_FLAG_BLANK_CH.

GAL_DATA_FLAG_SORTED_I
This bit has a value of 1 when the given dataset is sorted in an increasing manner. If this bit is 0 and GAL_DATA_FLAG_SORT_CH is 1, then the dataset has been checked and wasn’t sorted (increasing), so there is no more need for further checks.

GAL_DATA_FLAG_SORTED_D
This bit has a value of 1 when the given dataset is sorted in a decreasing manner. If this bit is 0 and GAL_DATA_FLAG_SORT_CH is 1, then the dataset has been checked and wasn’t sorted (decreasing), so there is no more need for further checks.

The macro GAL_DATA_FLAG_MAXFLAG contains the largest internally used bit-position. Higher-level flags can be defined with the bit-wise shift operators using this macro to define internal flags for libraries/programs that depend on Gnuastro without causing any possible conflict with the internal flags discussed above or having to check the values manually on every release.

int status
A context-specific status values for this data-structure. This integer will not be set by Gnuastro’s libraries. You can use it keep some additional information about the dataset (with integer constants) depending on your applications.

char *name
The name of the dataset. If the dataset is a multi-dimensional array and read/written as a FITS image, this will be the value in the EXTNAMESPACE FITS keyword. If the dataset is a one-dimensional table column, this will be the column name. If it is set to NULL (by default), it will be ignored.

char *unit
The units of the dataset (for example BUNIT in the standard FITS keywords) that will be read from or written to files/tables along with the dataset. If it is set to NULL (by default), it will be ignored.

char *comment
Any further explanation about the dataset which will be written to any output file if present.

disp_fmt Format to use for printing each element of the dataset to a plain text file, the acceptable values to this element are defined in Section 11.3.10 [Table input output (table.h)], page 568. Based on C’s printf standards.
disp_width
Width of printing each element of the dataset to a plain text file, the acceptable values to this element are defined in Section 11.3.10 [Table input output (table.h)], page 568. Based on C’s printf standards.

disp_precision
Precision of printing each element of the dataset to a plain text file, the acceptable values to this element are defined in Section 11.3.10 [Table input output (table.h)], page 568. Based on C’s printf standards.

gal_data_t *next
Through this pointer, you can link a gal_data_t with other datasets related datasets, for example the different columns in a dataset each have one gal_data_t associate with them and they are linked to each other using this element. There are several functions described below to facilitate using gal_data_t as a linked list. See Section 11.3.8 [Linked lists (list.h)], page 554, for more on these wonderful high-level constructs.

gal_data_t *block
Pointer to the start of the complete allocated block of memory. When this pointer is not NULL, the dataset is not treated as a contiguous patch of memory. Rather, it is seen as covering only a portion of the larger patch of memory that block points to. See Section 11.3.15 [Tessellation library (tile.h)], page 608, for a more thorough explanation and functions to help work with tiles that are created from this pointer.

11.3.6.2 Dataset allocation
Gnuastro’s main data container was defined in Section 11.3.6.1 [Generic data container (gal_data_t)], page 542. The functions listed in this section describe the most basic operations on gal_data_t: those related to allocation and freeing. These functions are declared in gnuastro/data.h which is also visible from the function names (see Section 11.3 [Gnuastro library], page 525).

gal_data_t *
gal_data_alloc (void *array, uint8_t type, size_t ndim, size_t *dsize, struct wcsprm *wcs, int clear, size_t minmapsize, int quietmmap, char *name, char *unit, char *comment)
Dynamically allocate a gal_data_t and initialize it with all the given values. See the description of gal_data_initialize and Section 11.3.6.1 [Generic data container (gal_data_t)], page 542, for more information. This function will often be the most frequently used because it allocates the gal_data_t hosting all the values and initializes it. Once you are done with the dataset, be sure to clean up all the allocated spaces with gal_data_free.

void *
gal_data_initialize (gal_data_t *data, void *array, uint8_t type, size_t ndim, size_t *dsize, struct wcsprm *wcs, int clear, size_t minmapsize, int quietmmap, char *name, char *unit, char *comment)
Initialize the given data structure (data) with all the given values. Note that the raw input gal_data_t must already have been allocated before calling this function.
For a description of each variable see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542. It will set the values and do the necessary allocations. If they aren’t NULL, all input arrays (dsize, wcs, name, unit, comment) are separately copied (allocated) by this function for usage in data, so you can safely use one value to initialize many datasets or use statically allocated variables in this function call. Once you are done with the dataset, you can free all the allocated spaces with gal_data_free_contents.

If array is not NULL, it will be directly copied into data->array (based on the total number of elements calculated from dsize) and no new space will be allocated for the array of this dataset, this has many low-level advantages and can be used to work on regions of a dataset instead of the whole allocated array (see the description under block in Section 11.3.6.1 [Generic data container (gal_data_t)], page 542, for one example). If the given pointer is not the start of an allocated block of memory or it is used in multiple datasets, be sure to set it to NULL (with data->array=NULL) before cleaning up with gal_data_free_contents.

ndim may be zero. In this case no allocation will occur, data->array and data->dsize will be set to NULL and data->size will be zero. However (when necessary) dsize must not have any zero values (a dimension of length zero is not defined).
initialize to fill each element of the array afterwards. The following code snippet is one example of doing this.

```c
size_t i;
gal_data_t *dataarr;
dataarr = gal_data_array_calloc(10);
for (i = 0; i < 10; ++i) gal_data_initialize(&dataarr[i], ...);
...
gal_data_array_free(dataarr, 10, 1);
```

**void**

`gal_data_array_free (gal_data_t *dataarr, size_t num, int free_array)`

Free all the `num` elements within `dataarr` and the actual allocated array. If `free_array` is not zero, then the array element of all the datasets will also be freed, see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542.

**gal_data_t **

`gal_data_array_ptr_calloc (size_t size)`

Allocate an array of pointers to Gnuastro’s generic data structure and initialize all pointers to NULL. This is useful when you want to allocate individual datasets later (for example with `gal_data_alloc`).

**void**

`gal_data_array_ptr_free (gal_data_t **dataptr, size_t size, int free_array)`;

Free all the individual datasets within the elements of `dataptr`, then free `dataptr` itself (the array of pointers that was probably allocated with `gal_data_array_ptr_calloc`).

### 11.3.6.4 Copying datasets

The functions in this section describes Gnuastro’s facilities to copy a given dataset into another. The new dataset can have a different type (including a string), it can be already allocated (in which case only the values will be written into it). In all these cases, if the input dataset is a tile or a list, only the data within the given tile, or the given node in a list, are copied. If the input is a list, the next pointer will also be copied to the output, see Section 11.3.8.9 [List of gal_data_t], page 566.

In many of the functions here, it is possible to copy the dataset to a new numeric data type (see Section 4.5 [Numeric data types], page 178. In such cases, Gnuastro's library is going to use the native conversion by C. So if you are converting to a smaller type, it is up to you to make sure that the values fit into the output type.

**gal_data_t **

`gal_data_copy (gal_data_t *in)`

Return a new dataset that is a copy of `in`, all of `in`’s meta-data will also copied into the output, except for block. If the dataset is a tile/list, only the given tile/node will be copied, the next pointer will also be copied however.

**gal_data_t **

`gal_data_copy_to_new_type (gal_data_t *in, uint8_t newtype)`

Return a copy of the dataset `in`, converted to `newtype`, see Section 11.3.3 [Library data types (type.h)], page 532, for Gnuastro library’s type identifiers. The returned
dataset will have all meta-data except their type and block equal to the input’s metadata. If the dataset is a tile/list, only the given tile/node will be copied, the next pointer will also be copied however.

**gal_data_t**
**gal_data_copy_to_new_type_free (gal_data_t *in, uint8_t newtype)**

Return a copy of the dataset in that is converted to newtype and free the input dataset. See Section 11.3.3 [Library data types (type.h)], page 532, for Gnuastro library’s type identifiers. The returned dataset will have all meta-data, except their type, equal to the input’s metadata (including next). Note that if the input is a tile within a larger block, it will not be freed. This function is similar to **gal_data_copy_to_new_type**, except that it will free the input dataset.

**void**
**gal_data_copy_to_allocated (gal_data_t *in, gal_data_t *out)**

Copy the contents of the array in into the already allocated array in out. The types of the input and output may be different, type conversion will be done internally. When in->size != out->size this function will behave as follows:

out->size < in->size

This function won’t re-allocate the necessary space, it will abort with an error, so please check before calling this function.

out->size > in->size

This function will write the values in out->size and out->dsize from the same values of in. So if you want to use a pre-allocated space/dataset multiple times with varying input sizes, be sure to reset out->size before every call to this function.

**gal_data_t**
**gal_data_copy_string_to_number (char *string)**

Read string into the smallest type that can store the value (see Section 4.5 [Numeric data types], page 178). This function is just a wrapper for the **gal_type_string_to_number**, but will put the value into a single-element dataset.

### 11.3.7 Dimensions (dimension.h)

An array is a contiguous region of memory. Hence, at the lowest level, every element of an array just has one single-valued position: the number of elements that lie between it and the first element in the array. This is also known as the index of the element within the array. A dataset’s number of dimensions is high-level abstraction (meta-data) that we project onto that contiguous patch of memory. When the array is interpreted as a one-dimensional dataset, this index is also the coordinate of the element. But once we associate the patch of memory with a higher dimension, there must also be one coordinate for each dimension.

The functions and macros in this section provide you with the tools to convert an index into a coordinate and vice-versa along with several other issues for example issues with the neighbors of an element in a multi-dimensional context.
size_t  
**gal_dimension_total_size**(size_t ndim, size_t *dsize)  
Return the total number of elements for a dataset with ndim dimensions that has dsize elements along each dimension.

int  
**gal_dimension_is_different**(gal_data_t *first, gal_data_t *second)  
Return 1 (one) if the two datasets don’t have the same size along all dimensions. This function will also return 1 when the number of dimensions of the two datasets are different.

size_t *  
**gal_dimension_increment**(size_t ndim, size_t *dsize)  
Return an allocated array that has the number of elements necessary to increment an index along every dimension. For example along the fastest dimension (last element in the dsize and returned arrays), the value is 1 (one).

size_t  
**gal_dimension_num_neighbors**(size_t ndim)  
The maximum number of neighbors (any connectivity) that a data element can have in ndim dimensions. Effectively, this function just returns $3^n - 1$ (where $n$ is the number of dimensions).

GAL_DIMENSION_FLT_TO_INT**(FLT)**  
[Function-like macro]  
Calculate the integer pixel position that the floating point FLT number belongs to. In the FITS format (and thus in Gnuastro), the center of each pixel is allocated on an integer (not it edge), so the pixel which hosts a floating point number cannot simply be found with internal type conversion.

void  
**gal_dimension_add_coords**(size_t *c1, size_t *c2, size_t *out, size_t ndim)  
For every dimension, add the coordinates in c1 with c2 and put the result into out. In other words, for dimension i run out[i]=c1[i]+c2[i];. Hence out may be equal to any one of c1 or c2.

size_t  
**gal_dimension_coord_to_index**(size_t ndim, size_t *dsize, size_t *coord)  
Return the index (counting from zero) from the coordinates in coord (counting from zero) assuming the dataset has ndim elements and the size of the dataset along each dimension is in the dsize array.

void  
**gal_dimension_index_to_coord**(size_t index, size_t ndim, size_t *dsize, size_t *coord)  
Fill in the coord array with the coordinates that correspond to index assuming the dataset has ndim elements and the size of the dataset along each dimension is in the dsize array. Note that both index and each value in coord are assumed to start from 0 (zero). Also that the space which coord points to must already be allocated before calling this function.
size_t

gal_dimension_dist_manhattan (size_t *a, size_t *b, size_t ndim)
Return the manhattan distance (see Wikipedia (https://en.wikipedia.org/wiki/Taxicab_geometry)) between the two coordinates a and b (each an array of ndim elements).

float

gal_dimension_dist_radial (size_t *a, size_t *b, size_t ndim)
Return the radial distance between the two coordinates a and b (each an array of ndim elements).

float

gal_dimension_dist_elliptical (double *center, double *pa_deg, double *q, size_t ndim, double *point)
Return the elliptical/ellipsoidal distance of the single point point (containing ndim values: coordinates of the point in each dimension) from an ellipse that is defined by center, pa_deg and q. center is the coordinates of the ellipse center (also with ndim elements). pa is the position-angle in degrees (the angle of the semi-major axis from the first dimension in a 2D ellipse) and q is the axis ratio.

In a 2D ellipse, pa and q are a single-element array. However, in a 3D ellipsoid, pa must have three elements, and q must have 2 elements. For more see Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440.

gal_data_t *

gal_dimension_collapse_sum (gal_data_t *in, size_t c_dim, gal_data_t *weight)
Collapse the input dataset (in) along the given dimension (c_dim, in C definition: starting from zero, from the slowest dimension), by summing all elements in that direction. If weight!=NULL, it must be a single-dimensional array, with the same size as the dimension to be collapsed. The respective weight will be multiplied to each element during the collapse.

For generality, the returned dataset will have a GAL_TYPE_FLOAT64 type. See Section 11.3.6.4 [Copying datasets], page 549, for converting the returned dataset to a desired type. Also, for more on the application of this function, see the Arithmetic program’s collapse-sum operator (which uses this function) in Section 6.2.3 [Arithmetic operators], page 276.

gal_data_t *

gal_dimension_collapse_mean (gal_data_t *in, size_t c_dim, gal_data_t *weight)
Similar to gal_dimension_collapse_sum (above), but the collapse will be done by calculating the mean along the requested dimension, not summing over it.

gal_data_t *

gal_dimension_collapse_number (gal_data_t *in, size_t c_dim)
Collapse the input dataset (in) along the given dimension (c_dim, in C definition: starting from zero, from the slowest dimension), by counting how many non-blank elements there are along that dimension.
For generality, the returned dataset will have a GAL_TYPE_INT32 type. See Section 11.3.6.4 [Copying datasets], page 549, for converting the returned dataset to a desired type. Also, for more on the application of this function, see the Arithmetic program’s collapse-number operator (which uses this function) in Section 6.2.3 [Arithmetic operators], page 276.

```c
[Function]
gal_data_t * gal_dimension_collapse_minmax (gal_data_t *in, size_t c_dim, int max1_min0)
```
Collapse the input dataset (in) along the given dimension (c_dim, in C definition: starting from zero, from the slowest dimension), by using the largest/smallest non-blank value along that dimension. If max1_min0 is non-zero, then the collapsed dataset will have the maximum value along the given dimension and if it is zero, the minimum.

```c
size_t gal_dimension_remove_extra (size_t ndim, size_t *dsize, struct wcsprm *wcs)
```
Remove extra dimensions (those that only have a length of 1) from the basic size information of a dataset. `ndim` is the number of dimensions and `dsize` is an array with `ndim` elements containing the size along each dimension in the C dimension order. When `wcs!=NULL`, the respective dimension will also be removed from the WCS. This function will return the new number of dimensions and the `dsize` elements will contain the length along each new dimension.

```c
GAL_DIMENSION_NEIGHBOR_OP (index, ndim, dsize, connectivity, dinc, operation)
```
Parse the neighbors of the element located at `index` and do the requested operation on them. This is defined as a macro to allow easy definition of any operation on the neighbors of a given element without having to use loops within your source code (the loops are implemented by this macro). For an example of using this function, please see Section 11.4.2 [Library demo - inspecting neighbors], page 663. The input arguments to this function-like macro are described below:

- `index` Distance of this element from the first element in the array on a contiguous patch of memory (starting from 0), see the discussion above.
- `ndim` The number of dimensions associated with the contiguous patch of memory.
- `dsize` The full array size along each dimension. This must be an array and is assumed to have the same number elements as `ndim`. See the discussion under the same element in Section 11.3.6.1 [Generic data container (gal_data_t)], page 542.
- `connectivity` Most distant neighbors to consider. Depending on the number of dimensions, different neighbors may be defined for each element. This function-like macro distinguish between these different neighbors with this argument. It has a value between 1 (one) and `ndim`. For example in a 2D dataset, 4-connected neighbors have a connectivity of 1 and 8-connected neighbors have a connectivity of 2. Note that this is inclusive, so in this example, a connectivity of 2 will also include connectivity 1 neighbors.
dinc  An array keeping the length necessary to increment along each dimension. You can make this array with the following function. Just don’t forget to free the array after you are done with it:

```c
size_t *dinc=gal_dimension_increment(ndim, dsize);
```

dinc depends on ndim and dsize, but it must be defined outside this function-like macro since it involves allocation to help in performance.

**operation**

Any C operation that you would like to do on the neighbor. This macro will provide you a nind variable that can be used as the index of the neighbor that is currently being studied. It is defined as ‘size_t ndim;’. Note that operation will be repeated the number of times there is a neighbor for this element.

This macro works fully within its own {} block and except for the nind variable that shows the neighbor’s index, all the variables within this macro’s block start with gdn_.

### 11.3.8 Linked lists (list.h)

An array is a contiguous region of memory that is very efficient and easy to use for recording and later accessing any random element as fast as any other. This makes array the primary data container when you have many elements (for example an image which has millions of pixels). One major problem with an array is that the number of elements that go into it must be known in advance and adding or removing an element will require a re-set of all the other elements. For example if you want to remove the 3rd element in a 1000 element array, all 997 subsequent elements have to pulled back by one position, the reverse will happen if you need to add an element.

In many contexts such situations never come up, for example you don’t want to shift all the pixels in an image by one or two pixels from some random position in the image: their positions have scientific value. But in other contexts you will find your self frequently adding/removing an a-priori unknown number of elements. Linked lists (or lists for short) are the data-container of choice in such situations. As in a chain, each node in a list is an independent C structure, keeping its own data along with pointer(s) to its immediate neighbor(s). Below, you can see one simple linked list node structure along with an ASCII art schematic of how we can use the next pointer to add any number of elements to the list that we want. By convention, a list is terminated when next is the NULL pointer.

```c
struct list_float /* --------- --------- */
{ /* | Value | | Value | */
    float value; /* | --- | | --- | */
    struct list_float *next; /* | next-|--> | next-|--> NULL */
} /* --------- --------- */
```

The schematic shows another great advantage of linked lists: it is very easy to add or remove/pop a node anywhere in the list. If you want to modify the first node, you just have to change one pointer. If it is in the middle, you just have to change two. You initially define a variable of this type with a NULL pointer as shown below:

```c
struct list_float *mylist=NULL;
```

To add or remove/pop a node from the list you can use functions provided for the respective type in the sections below.
When you add an element to the list, it is conventionally added to the “top” of the list: the general list pointer will point to the newly created node, which will point to the previously created node and so on. So when you “pop” from the top of the list, you are actually retrieving the last value you put in and changing the list pointer to the next youngest node. This is thus known as a “last-in-first-out” list. This is the most efficient type of linked list (easier to implement and faster to process). Alternatively, you can add each newly created node at the end of the list. If you do that, you will get a “first-in-first-out” list. But that will force you to go through the whole list for each new element that is created (this will slow down the processing)\(^\text{18}\).

The node example above creates the simplest kind of a list. We can define each node with two pointers to both the next and previous neighbors, this is called a “Doubly linked list”. In general, lists are very powerful and simple constructs that can be very useful. But going into more detail would be out of the scope of this short introduction in this book. Wikipedia (https://en.wikipedia.org/wiki/Linked_list) has a nice and more thorough discussion of the various types of lists. To appreciate/use the beauty and elegance of these powerful constructs even further, see Chapter 2 (Information Structures, in volume 1) of Donald Knuth’s “The art of computer programming”.

In this section we will review the functions and structures that are available in Gnuastro for working on lists. They differ by the type of data that each node can keep. For each linked-list node structure, we will first introduce the structure, then the functions for working on the structure. All these structures and functions are defined and declared in gnuastro/list.h.

11.3.8.1 List of strings

Probably one of the most common lists you will be using are lists of strings. They are the best tools when you are reading the user’s inputs, or when adding comments to the output files. Below you can see Gnuastro’s string list type and several functions to help in adding, removing/popping, reversing and freeing the list.

\[ \text{gal_list_str_t} \]  
Type (C struct)

A single node in a list containing a string of characters.

\[
\text{typedef struct gal_list_str_t} \\
\{ \\
\text{char *v; } \\
\text{struct gal_list_str_t *next; } \\
\} \text{gal_list_str_t;}
\]

\[ \text{void} \] [Function]

\[
\text{gal_list_str_add (gal_list_str_t **list, char *value, int allocate)}
\]

Add a new node to the list of strings (list) and update it. The new node will contain the string value. If allocate is not zero, space will be allocated specifically for the string of the new node and the contents of value will be copied into it. This can be useful when your string may be changed later in the program, but you want your list

---

\(^{18}\) A better way to get a first-in-first-out is to first keep the data as last-in-first-out until they are all read. Afterwards, reverse the list by popping each node and immediately add it to the new list. This practically reverses the last-in-first-out list to a first-in-first-out one. All the list types discussed in this chapter have a function with a \_reverse suffix for this job.
Chapter 11: Library

Here is one short/simple example of initializing and adding elements to a string list:

```c
gal_list_str_t *strlist=NULL;
gal_list_str_add(&strlist, "bottom of list.");
gal_list_str_add(&strlist, "second last element of list.");
```

```c
char *

gal_list_str_pop (gal_list_str_t **list)
Pop the top element of list, change list to point to the next node in the list, and return the string that was in the popped node. If *list==NULL, then this function will also return a NULL pointer.

size_t

gal_list_str_number (gal_list_str_t *list)
Return the number of nodes in list.

size_t

gal_list_str_last (gal_list_str_t *list)
Return a pointer to the last node in list.

void

gal_list_str_print (gal_list_str_t *list)
Print the strings within each node of *list on the standard output in the same order that they are stored. Each string is printed on one line. This function is mainly good for checking/debugging your program. For program outputs, its best to make your own implementation with a better, more user-friendly, format. For example the following code snippet.

```c
size_t i;
gal_list_str_t *tmp;
for(tmp=list; tmp!=NULL; tmp=tmp->next)
printf("String %zu: %s\n", i, tmp->v);
```

void

gal_list_str_reverse (gal_list_str_t **list)
Reverse the order of the list such that the top node in the list before calling this function becomes the bottom node after it.

void

gal_list_str_free (gal_list_str_t *list, int freevalue)
Free every node in list. If freevalue is not zero, also free the string within the nodes.

### 11.3.8.2 List of int32_t

Signed integers are the best types when you are dealing with a positive or negative integers. They are generally useful in many contexts, for example when you want to keep the order of a series of states (each state stored as a given number in an `enum` for example). On many modern systems, `int32_t` is just an alias for `int`, so you can use them interchangeably. To make sure, check the size of `int` on your system:
gal_list_i32_t

[Type (C struct)]
A single node in a list containing a 32-bit signed integer (see Section 4.5 [Numeric data types], page 178).

typedef struct gal_list_i32_t {
    int32_t v;
    struct gal_list_i32_t *next;
} gal_list_i32_t;

void gal_list_i32_add (gal_list_i32_t **list, int32_t value)
Add a new node (containing value) to the top of the list of int32_t s (uint32_t is equal to int on many modern systems), and update list. Here is one short example of initializing and adding elements to a string list:

    gal_list_i32_t *i32list=NULL;
    gal_list_i32_add(&i32list, 52);
    gal_list_i32_add(&i32list, -4);

int32_t gal_list_i32_pop (gal_list_i32_t **list)
Pop the top element of list and return the value. This function will also change list to point to the next node in the list. If *list==NULL, then this function will also return GAL_BLANK_INT32 (see Section 11.3.5 [Library blank values (blank.h)], page 538).

size_t gal_list_i32_number (gal_list_i32_t *list)
Return the number of nodes in list.

size_t gal_list_i32_last (gal_list_i32_t *list)
Return a pointer to the last node in list.

void gal_list_i32_print (gal_list_i32_t *list)
Print the integers within each node of *list on the standard output in the same order that they are stored. Each integer is printed on one line. This function is mainly good for checking/debugging your program. For program outputs, its best to make your own implementation with a better, more user-friendly format. For example the following code snippet. You can also modify it to print all values in one line, etc, depending on the context of your program.

    size_t i;
    gal_list_i32_t *tmp;
    for(tmp=list; tmp!=NULL; tmp=tmp->next)
        printf("String %zu: %s\n", i, tmp->v);
Chapter 11: Library

void gal_list_i32_reverse (gal_list_i32_t **list)
    Reverse the order of the list such that the top node in the list before calling this
    function becomes the bottom node after it.

int32_t * gal_list_i32_to_array (gal_list_i32_t *list, int reverse, size_t *num)
    Dynamically allocate an array and fill it with the values in list. The function will
    return a pointer to the allocated array and put the number of elements in the array
    into the num pointer. If reverse has a non-zero value, the array will be filled in the
    opposite order of elements in list. This function can be useful after you have finished
    reading an initially unknown number of values and want to put them in an array for
    easy random access.

void gal_list_i32_free (gal_list_i32_t *list)
    Free every node in list.

11.3.8.3 List of size_t

The size_t type is a unique type in C: as the name suggests it is defined to store sizes,
or more accurately, the distances between memory locations. Hence it is always positive
(an unsigned type) and it is directly related to the address-able spaces on the host system:
on 32-bit and 64-bit systems it is an alias for uint32_t and uint64_t, respectively (see
Section 4.5 [Numeric data types], page 178).

size_t is the default compiler type to index an array (recall that an array index in
C is just a pointer increment of a given size). Since it is unsigned, its a great type for
counting (where negative is not defined), you are always sure it will never exceed the
system’s (virtual) memory and since its name has the word “size” inside it, it provides
a good level of documentation. In Gnuastro, we do all counting and array indexing with
this type, so this list is very handy. As discussed above, size_t maps to different types on
different machines, so a portable way to print them with printf is to use C99’s %zu format.

gal_list_sizet_t
    A single node in a list containing a size_t value (which maps to uint32_t or uint64_
t on 32-bit and 64-bit systems), see Section 4.5 [Numeric data types], page 178.
    
    typedef struct gal_list_sizet_t
    {
        size_t v;
        struct gal_list_sizet_t *next;
    } gal_list_sizet_t;

void gal_list_sizet_add (gal_list_sizet_t **list, size_t value)
    Add a new node (containing value) to the top of the list of size_t's and update
    list. Here is one short example of initializing and adding elements to a string list:
    
    gal_list_sizet_t *slist=NULL;

19 So you know that a variable of this type is not used to store some generic state for example.
gal_list_sizet_add(&slist, 45493);
gal_list_sizet_add(&slist, 930484);

sizet_t  

[Function]

gal_list_sizet_pop (gal_list_sizet_t **list)
Pop the top element of list and return the value. This function will also change list to point to the next node in the list. If *list==NULL, then this function will also return GAL_BLANK_SIZE_T (see Section 11.3.5 [Library blank values (blank.h)], page 538).

size_t  

[Function]

gal_list_sizet_number (gal_list_sizet_t *list)
Return the number of nodes in list.

size_t  

[Function]

gal_list_sizet_last (gal_list_sizet_t *list)
Return a pointer to the last node in list.

void  

[Function]

gal_list_sizet_print (gal_list_sizet_t *list)
Print the values within each node of *list on the standard output in the same order that they are stored. Each integer is printed on one line. This function is mainly good for checking/debugging your program. For program outputs, its best to make your own implementation with a better, more user-friendly format. For example, the following code snippet. You can also modify it to print all values in one line, etc, depending on the context of your program.

```c
size_t i;
gal_list_sizet_t *tmp;
for(tmp=list; tmp!=NULL; tmp=tmp->next)
    printf("String %zu: %zu\n", i, tmp->v);
```

void  

[Function]

gal_list_sizet_reverse (gal_list_sizet_t **list)
Reverse the order of the list such that the top node in the list before calling this function becomes the bottom node after it.

size_t *  

[Function]

gal_list_sizet_to_array (gal_list_sizet_t *list, int reverse, size_t *num)
Dynamically allocate an array and fill it with the values in list. The function will return a pointer to the allocated array and put the number of elements in the array into the num pointer. If reverse has a non-zero value, the array will be filled in the inverse of the order of elements in list. This function can be useful after you have finished reading an initially unknown number of values and want to put them in an array for easy random access.

void  

[Function]

gal_list_sizet_free (gal_list_sizet_t *list)
Free every node in list.
11.3.8.4 List of float

Single precision floating point numbers can accurately store real number until 7.2 decimals and only consume 4 bytes (32-bits) of memory, see Section 4.5 [Numeric data types], page 178. Since astronomical data rarely reach that level of precision, single precision floating points are the type of choice to keep and read data. However, when processing the data, it is best to use double precision floating points (since errors propagate).

gal_list_f32_t  
[Type (C struct)]  A single node in a list containing a 32-bit single precision float value: see Section 4.5 [Numeric data types], page 178.

typedef struct gal_list_f32_t
{
    float v;
    struct gal_list_f32_t *next;
} gal_list_f32_t;

void  
[Function]  gal_list_f32_add (gal_list_f32_t **list, float value)  
Add a new node (containing value) to the top of the list of floats and update list. Here is one short example of initializing and adding elements to a string list:

gal_list_f32_t *flist=NULL;
gal_list_f32_add(&flist, 3.89);
gal_list_f32_add(&flist, 1.23e-20);

float  
[Function]  gal_list_f32_pop (gal_list_f32_t **list)  
Pop the top element of list and return the value. This function will also change list to point to the next node in the list. If *list==NULL, then this function will return GAL_BLANK_FLOAT32 (NaN, see Section 11.3.5 [Library blank values (blank.h)], page 538).

size_t  
[Function]  gal_list_f32_number (gal_list_f32_t *list)  
Return the number of nodes in list.

size_t  
[Function]  gal_list_f32_last (gal_list_f32_t *list)  
Return a pointer to the last node in list.

void  
[Function]  gal_list_f32_print (gal_list_f32_t *list)  
Print the values within each node of *list on the standard output in the same order that they are stored. Each floating point number is printed on one line. This function is mainly good for checking/debugging your program. For program outputs, its best to make your own implementation with a better, more user-friendly format. For example, in the following code snippet. You can also modify it to print all values in one line, etc, depending on the context of your program.

    size_t i;
Chapter 11: Library

```c
gal_list_f32_t *tmp;
for(tmp=list; tmp!=NULL; tmp=tmp->next)
    printf("Node %zu: %f\n", i, tmp->v);
```

**Function**

```c
void gal_list_f32_reverse (gal_list_f32_t **list)
    Reverse the order of the list such that the top node in the list before calling this function becomes the bottom node after it.
```

**Function**

```c
float * gal_list_f32_to_array (gal_list_f32_t *list, int reverse, size_t *num)
    Dynamically allocate an array and fill it with the values in list. The function will return a pointer to the allocated array and put the number of elements in the array into the num pointer. If reverse has a non-zero value, the array will be filled in the inverse of the order of elements in list. This function can be useful after you have finished reading an initially unknown number of values and want to put them in an array for easy random access.
```

**Function**

```c
void gal_list_f32_free (gal_list_f32_t *list)
    Free every node in list.
```

### 11.3.8.5 List of double

Double precision floating point numbers can accurately store real number until 15.9 decimals and consume 8 bytes (64-bits) of memory, see Section 4.5 [Numeric data types], page 178. This level of precision makes them very good for serious processing in the middle of a program's execution: in many cases, the propagation of errors will still be insignificant compared to actual observational errors in a data set. But since they consume 8 bytes and more CPU processing power, they are often not the best choice for storing and transferring of data.

```c
gal_list_f64_t
    A single node in a list containing a 64-bit double precision double value: see Section 4.5 [Numeric data types], page 178.

    typedef struct gal_list_f64_t
    {
        double v;
        struct gal_list_f64_t *next;
    } gal_list_f64_t;
```

**Function**

```c
void gal_list_f64_add (gal_list_f64_t **list, double value)
    Add a new node (containing value) to the top of the list of doubles and update list. Here is one short example of initializing and adding elements to a string list:
    gal_list_f64_t *dlist=NULL;
    gal_list_f64_add(&dlist, 3.8129395763193);
    gal_list_f64_add(&dlist, 1.239378923931e-20);
```
double
gal_list_f64_pop (gal_list_f64_t **list)
    Pop the top element of list and return the value. This function will also change
    list to point to the next node in the list. If *list==NULL, then this function will
    return GAL_BLANK_FLOAT64 (NaN, see Section 11.3.5 [Library blank values (blank.h)],
    page 538).

size_t
gal_list_f64_number (gal_list_f64_t *list)
    Return the number of nodes in list.

size_t
gal_list_f64_last (gal_list_f64_t *list)
    Return a pointer to the last node in list.

void
gal_list_f64_print (gal_list_f64_t *list)
    Print the values within each node of *list on the standard output in the same order
    that they are stored. Each floating point number is printed on one line. This function
    is mainly good for checking/debugging your program. For program outputs, its best
    to make your own implementation with a better, more user-friendly format. For
    example, in the following code snippet. You can also modify it to print all values in
    one line, etc, depending on the context of your program.
    
    size_t i;
    gal_list_f64_t *tmp;
    for(tmp=list; tmp!=NULL; tmp=tmp->next)
        printf("Node %zu: %f\n", i, tmp->v);

void
gal_list_f64_reverse (gal_list_f64_t **list)
    Reverse the order of the list such that the top node in the list before calling this
    function becomes the bottom node after it.

double *
gal_list_f64_to_array (gal_list_f64_t *list, int reverse, size_t *num)
    Dynamically allocate an array and fill it with the values in list. The function will
    return a pointer to the allocated array and put the number of elements in the array
    into the num pointer. If reverse has a non-zero value, the array will be filled in the
    inverse of the order of elements in list. This function can be useful after you have
    finished reading an initially unknown number of values and want to put them in an
    array for easy random access.

void
gal_list_f64_free (gal_list_f64_t *list)
    Free every node in list.

11.3.8.6 List of void *

In C, void * is the most generic pointer. Usually pointers are associated with the type of
content they point to. For example int * means a pointer to an integer. This ancillary
information about the contents of the memory location is very useful for the compiler, catching bad errors and also documentation (it helps the reader see what the address in memory actually contains). However, void * is just a raw address (pointer), it contains no information on the contents it points to.

These properties make the void * very useful when you want to treat the contents of an address in different ways. You can use the void * list defined in this section and its function on any kind of data: for example you can use it to keep a list of custom data structures that you have built for your own separate program. Each node in the list can keep anything and this gives you great versatility. But in using void *, please beware that “with great power comes great responsibility”.

gal_list_void_t
   [Type (C struct)]

   A single node in a list containing a void * pointer.
   typedef struct gal_list_void_t
   {
      void *v;
      struct gal_list_void_t *next;
   } gal_list_void_t;

void
   [Function]

gal_list_void_add (gal_list_void_t **list, void *value)
   Add a new node (containing value) to the top of the list of void *s and update list. Here is one short example of initializing and adding elements to a string list:

   gal_list_void_t *vlist=NULL;
   gal_list_f64_add(&vlist, some_pointer);
   gal_list_f64_add(&vlist, another_pointer);

void *
   [Function]

gal_list_void_pop (gal_list_void_t **list)
   Pop the top element of list and return the value. This function will also change list to point to the next node in the list. If *list==NULL, then this function will return NULL.

size_t
   [Function]

gal_list_void_number (gal_list_void_t *list)
   Return the number of nodes in list.

size_t
   [Function]

gal_list_void_last (gal_list_void_t *list)
   Return a pointer to the last node in list.

void
   [Function]

gal_list_void_reverse (gal_list_void_t **list)
   Reverse the order of the list such that the top node in the list before calling this function becomes the bottom node after it.

void
   [Function]

gal_list_void_free (gal_list_void_t *list)
   Free every node in list.
11.3.8.7 Ordered list of size_t

Positions/sizes in a dataset are conventionally in the size_t type (see Section 11.3.8.3 [List of size_t, page 558] and it sometimes occurs that you want to parse and read the values in a specific order. For example you want to start from one pixel and add pixels to the list based on their distance to that pixel. So that ever time you pop an element from the list, you know it is the nearest that has not yet been studied. The gal_list_osizet_t type and its functions in this section are designed to facilitate such operations.

```c
typedef struct gal_list_osizet_t
{
    size_t v; /* The actual value. */
    float s; /* The parameter to sort by. */
    struct gal_list_osizet_t *next;
} gal_list_osizet_t;
```

**void** gal_list_osizet_add (gal_list_osizet_t **list, size_t value, float tosort)

Allocate space for a new node in list, and store value and tosort into it. The new node will not necessarily be at the “top” of the list. If *list!=NULL, then the tosort values of existing nodes is inspected and the given node is placed in the list such that the top element (which is popped with **gal_list_osizet_pop** has the smallest tosort value.

```c
size_t gal_list_osizet_pop (gal_list_osizet_t **list, float *sortvalue)
Pop a node from the top of list, return the node’s value and put its sort value in the space that sortvalue points to. This function will also free the allocated space for the popped node and after this function, list will point to the next node (which has a larger tosort element).
```

**void** gal_list_osizet_to_sizet_free (gal_list_osizet_t *in, gal_list_sizet_t **out)

Convert the ordered list of size_ts into an ordinary size_t linked list. This can be useful when all the elements have been added and you just need to pop-out elements and don’t care about the sorting values any more. After the conversion is done, this function will free the input list. Note that the out list doesn’t have to be empty. If it already contains some nodes, the new nodes will be added on top of them.

11.3.8.8 Doubly linked ordered list of size_t

An ordered list of indices is required in many contexts, one example was discussed at the beginning of Section 11.3.8.7 [Ordered list of size_t, page 564]. But the list that was introduced there only has one point of entry: you can always only parse the list from
smallest to largest. In this section, the doubly-linked `gal_list_dosizet_t` node is defined which will allow us to parse the values in ascending or descending order.

```
gal_list_dosizet_t  [Type (C struct)]
 Doubly-linked, ordered size_t list node structure. Each node in this Doubly-linked list contains a size_t value and a floating point value. The floating point value is used as a reference to add new nodes in a sorted manner. In the functions here, this linked list can be pointed to by two pointers (largest and smallest) with the following format:
```
```
largest pointer
| NULL <-- (v0,s0) <-- (v1,s1) <-- ... (vn,sn) --> NULL |

smallest pointer
```
At any moment, the two pointers will point to the nodes containing the “largest” and “smallest” values and the rest of the nodes will be sorted. This is useful when an unknown number of nodes are being added continuously and during the operations it is important to have the nodes in a sorted format.

```
typedef struct gal_list_dosizet_t
{
    size_t v; /* The actual value. */
    float s; /* The parameter to sort by. */
    struct gal_list_dosizet_t *prev;
    struct gal_list_dosizet_t *next;
} gal_list_dosizet_t;
```

```
void  [Function]
gal_list_dosizet_add (gal_list_dosizet_t **largest, gal_list_dosizet_t **smallest, size_t value, float tosort)
 Allocate space for a new node in list, and store value and tosort into it. If the list is empty, both largest and smallest must be NULL.
```
```
size_t  [Function]
gal_list_dosizet_pop_smallest (gal_list_dosizet_t **largest,
               gal_list_dosizet_t **smallest, float tosort)
 Pop the value with the smallest reference from the doubly linked list and store the reference into the space pointed to by tosort. Note that even though only the smallest pointer will be popped, when there was only one node in the list, the largest pointer also has to change, so we need both.
```
```
void  [Function]
gal_list_dosizet_print (gal_list_dosizet_t *largest, gal_list_dosizet_t *
   *smallest)
 Print the largest and smallest values sequentially until the list is parsed.
```
```
void  [Function]
gal_list_dosizet_to_sizet (gal_list_dosizet_t *in, gal_list_sizet_t **out)
 Convert the doubly linked, ordered size_t list into a singly-linked list of size_t.
```
void

gal_list_dosizet_free (gal_list_dosizet_t *largest)
Free the doubly linked, ordered sizet_t list.

11.3.8.9 List of gal_data_t

Gnuastro’s generic data container has a next element which enables it to be used as a
singly-linked list (see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542).
The ability to connect the different data containers offers great advantages. For example
each column in a table in an independent dataset: with its own name, units, numeric data
type (see Section 4.5 [Numeric data types], page 178). Another application is in Tessellating
an input dataset into separate tiles or only studying particular regions, or tiles, of a larger
dataset (see Section 4.8 [Tessellation], page 189, and Section 11.3.15 [Tessellation library
(tile.h)], page 608). Each independent tile over the dataset can be connected to the others
as a linked list and thus any number of tiles can be represented with one variable.

void

gal_list_data_add (gal_data_t **list, gal_data_t *newnode)
Add an already allocated dataset (newnode) to top of list. Note that if
newnode->next!=NULL (newnode is itself a list), then list will be added to its end.

In this example multiple images are linked together as a list:

```c
int quietmmap=1;
size_t minmapsize=-1;
gal_data_t *tmp, *list=NULL;
tmp = gal_fits_img_read("file1.fits", "1", minmapsize, quietmmap);
gal_list_data_add( &list, tmp );
tmp = gal_fits_img_read("file2.fits", "1", minmapsize, quietmmap);
gal_list_data_add( &list, tmp );
```

void

gal_list_data_add_alloc (gal_data_t **list, void *array, uint8_t type, size_t
ndim, size_t *dsize, struct wcsprm *wcs, int clear, size_t minmapsize,
int quietmmap, char *name, char *unit, char *comment)
Allocate a new dataset (with gal_data_alloc in Section 11.3.6.2 [Dataset allocation],
page 547) and put it as the first element of list. Note that if this is the first node
to be added to the list, list must be NULL.

void

gal_data_t *
gal_list_data_pop (gal_data_t **list)
Pop the top node from list and return it.

void

gal_list_data_reverse (gal_data_t **list)
Reverse the order of the list such that the top node in the list before calling this
function becomes the bottom node after it.

void

gal_data_t **
gal_list_data_to_array_ptr (gal_data_t *list, size_t *num)
Allocate and return an array of gal_data_t * pointers with the same number of
elements as the nodes in list. The pointers will be put in the same order that the
list is parsed. Hence the N-th element in the array will point to the same dataset that the N-th node in the list points to.

```c
size_t gal_list_data_number (gal_data_t *list)
  Return the number of nodes in list.
size_t gal_list_data_last (gal_data_t *list)
  Return a pointer to the last node in list.
void gal_list_data_free (gal_data_t *list)
  Free all the datasets in list along with all the allocated spaces in each.
```

### 11.3.9 Array input output

Getting arrays (commonly images or cubes) from a file into your program or writing them after the processing into an output file are some of the most common operations. The functions in this section are designed for such operations with the known file types. The functions here are thus just wrappers around functions of lower-level file type functions of this library, for example Section 11.3.11 [FITS files (fits.h)], page 572, or Section 11.3.12.2 [TIFF files (tiff.h)], page 590. If the file type of the input/output file is already known, you can use the functions in those sections respectively.

```c
int gal_array_name_recognized (char *filename)
  Return 1 if the given file name corresponds to one of the recognized file types for reading arrays.
int gal_array_name_recognized_multiext (char *filename)
  Return 1 if the given file name corresponds to one of the recognized file types for reading arrays which may contain multiple extensions (for example FITS or TIFF) formats.
int gal_array_file_recognized (char *filename)
  Similar to gal_array_name_recognized, but for FITS files, it will also check the contents of the file if the recognized file name suffix is not found. See the description of gal_fits_file_recognized for more (Section 11.3.11.1 [FITS Macros, errors and filenames], page 573).

gal_data_t gal_array_read (char *filename, char *extension, gal_list_str_t *lines, size_t minmapsize, int quietmmap)
  Read the array within the given extension (extension) of filename, or the lines list (see below). If the array is larger than minmapsize bytes, then it won't be read into RAM, but a file on the HDD/SSD (no difference for the programmer). Messages about the memory-mapped file can be disabled with quietmmap.
```
extension will be ignored for files that don’t support them (for example JPEG or text). For FITS files, extension can be a number or a string (name of the extension), but for TIFF files, it has to be number. In both cases, counting starts from zero.

For multi-channel formats (like RGB images in JPEG or TIFF), this function will return a Section 11.3.8.9 [List of gal_data_t], page 566: one data structure per channel. Thus if you just want a single array (and want to check if the user hasn’t given a multi-channel input), you can check the next pointer of the returned gal_data_t.

lines is a list of strings with each node representing one line (including the new-line character), see Section 11.3.8.1 [List of strings], page 555. It will mostly be the output of gal_txt_stdin_read, which is used to read the program’s input as separate lines from the standard input (see Section 11.3.12.1 [Text files (txt.h)], page 587). Note that filename and lines are mutually exclusive and one of them must be NULL.

```
void [Function]
gal_array_read_to_type (char *filename, char *extension, gal_list_str_t *
lines, uint8_t type, size_t minmapsize, int quietmmap)
Similar to gal_array_read, but the output data structure(s) will have a numeric
data type of type, see Section 4.5 [Numeric data types], page 178.
```

```
void [Function]
gal_array_read_one_ch (char *filename, char *extension, gal_list_str_t *
lines, size_t minmapsize, int quietmmap)
Read the dataset within filename (extension/hdu/dir extension) and make sure it is only a single channel. This is just a simple wrapper around gal_array_read that checks if there was more than one dataset and aborts with an informative error if there is more than one channel in the dataset.

Formats like JPEG or TIFF support multiple channels per input, but it may happen that your program only works on a single dataset. This function can be a convenient way to make sure that the data that comes into your program is only one channel.
```

```
void [Function]
gal_array_read_one_ch_to_type (char *filename, char *extension,
gal_list_str_t *lines, uint8_t type, size_t minmapsize, int quietmmap)
Similar to gal_array_read_one_ch, but the output data structure will have a numeric
data type of type, see Section 4.5 [Numeric data types], page 178.
```

11.3.10 Table input output (table.h)

Tables are a collection of one dimensional datasets that are packed together into one file. They are the single most common format to store high-level (processed) information, hence they play a very important role in Gnuastro. For a more thorough introduction, please see Section 5.3 [Table], page 230. Gnuastro’s Table program, and all the other programs that can read from and write into tables, use the functions of this section for reading and writing their input/output tables. For a simple demonstration of using the constructs introduced here, see Section 11.4.4 [Library demo - reading and writing table columns], page 667.

Currently only plain text (see Section 4.7.2 [Gnuastro text table format], page 185) and FITS (ASCII and binary) tables are supported by Gnuastro. However, the low-level table
infra-structure is written such that accommodating other formats is also possible and in future releases more formats will hopefully be supported. Please don’t hesitate to suggest your favorite format so it can be implemented when possible.

The default width and precision for generic types to use in writing numeric types into a text file (plain text and FITS ASCII tables). When the dataset doesn’t have any pre-set width and precision (see `disp_width` and `disp_precision` in Section 11.3.6.1 [Generic data container (gal_data_t)], page 542) these will be directly used in C’s `printf` command to write the number as a string.

The display format used in C’s `printf` to display data of different types. The _STRING and _DECIMAL are unique for printing strings and signed integers, they are mainly here for completeness. However, unsigned integers and floating points can be displayed in multiple formats:

**Unsigned integer**

For unsigned integers, it is possible to choose from _UDECIMAL (unsigned decimal), _OCTAL (octal notation, for example 125 in decimal will be displayed as 175), and _HEX (hexadecimal notation, for example 125 in decimal will be displayed as 7D).

**Floating point**

For floating point, it is possible to display the number in _FLOAT (floating point, for example 1500.345), _EXP (exponential, for example 1.500345e+03), or _GENERAL which is the best of the two for the given number.

All the current acceptable table formats to Gnuastro. The AFITS and BFITS represent FITS ASCII tables and FITS Binary tables. You can use these anywhere you see the `tableformat` variable.
GAL_TABLE_SEARCH_INVALID  [Macro]
GAL_TABLE_SEARCH_NAME  [Macro]
GAL_TABLE_SEARCH_UNIT  [Macro]
GAL_TABLE_SEARCH_COMMENT  [Macro]

When the desired column is not a number, these values determine if the string to match, or regular expression to search, be in the name, units or comments of the column meta data. These values should be used for the searchin variables of the functions.

`gal_data_t * gal_table_info (char *filename, char *hdu, gal_list_str_t *lines, size_t *numcols, size_t *numrows, int *tableformat)`

Store the information of each column of a table into an array of data structures with numcols datasets (one data structure for each column). The number of rows is stored in numrows. The format of the table (e.g., ASCII text file, or FITS binary or ASCII table) will be put in tableformat (macros defined above). If the filename is not a FITS file, then hdu will not be used (can be NULL).

The input must be either a file (specified by filename) or a list of strings (lines). lines is a list of strings with each node representing one line (including the new-line character), see Section 11.3.8.1 [List of strings], page 555. It will mostly be the output of gal_txt_stdin_read, which is used to read the program’s input as separate lines from the standard input (see Section 11.3.12.1 [Text files (txt.h)], page 587). Note that filename and lines are mutually exclusive and one of them must be NULL.

In the output datasets, only the meta-data strings (column name, units and comments), will be allocated and set. This function is just for column information (metadata), not column contents.

`void gal_table_print_info (gal_data_t *allcols, size_t numcols, size_t numrows)`

This program will print the column information for all the columns (output of `gal_table_info`). The output is in the same format as this command with Gnuastro Table program (see Section 5.3 [Table], page 230):

```
$ asttable --info table.fits
```

`gal_data_t * gal_table_read (char *filename, char *hdu, gal_list_str_t *lines, gal_list_str_t *cols, int searchin, int ignorecase, size_t numthreads, size_t minmapsize, int quietmmap, size_t *colmatch)`

Read the specified columns in a file (named filename), or list of strings (lines) into a linked list of data structures. If the file is FITS, then hdu will also be used, otherwise, hdu is ignored.

lines is a list of strings with each node representing one line (including the new-line character), see Section 11.3.8.1 [List of strings], page 555. It will mostly be the output of gal_txt_stdin_read, which is used to read the program’s input as separate lines from the standard input (see Section 11.3.12.1 [Text files (txt.h)], page 587). Note that filename and lines are mutually exclusive and one of them must be NULL.
The information to search for columns should be specified by the cols list of strings (see Section 11.3.8.1 [List of strings], page 555). The string in each node of the list may be a number, an exact match to a column name, or a regular expression (in GNU AWK format) enclosed in / /. The searchin value must be one of the macros defined above. If cols is NULL, then this function will read the full table.

For FITS tables, each column will be read independently. Therefore they will be read in numthreads CPU threads to greatly speed up the reading when there are many columns and rows. However, this only happens if CFITSIO was configured with --enable-reentrant. This test has been done at Gnuastro’s configuration time; if so, GAL_CONFIG_HAVE_FITS_IS_REENTRANT will have a value of 1, otherwise, it will have a value of 0. For more on this macro, see Section 11.3.1 [Configuration information (config.h)], page 526. Multi-threaded table reading is not currently applicable to other table formats (only for FITS tables).

The output is an individually allocated list of datasets (see Section 11.3.8.9 [List of gal_data_t], page 566) with the same order of the cols list. Note that one column node in the cols list might give multiple columns (for example from regular expressions), in this case, the order of output columns that correspond to that one input, are in order of the table (which column was read first). So the first requested column is the first popped data structure and so on.

if colmatch!=NULL, it is assumed to be an array that has at least the same number of elements as nodes in the cols list. The number of columns that matched each input column will be stored in each element.

gal_list_sizet_t * gal_table_list_of_indexs (gal_list_str_t *cols, gal_data_t *allcols, size_t numcols, int searchin, int ignorecase, char *filename, char *hdu, size_t *colmatch)

Returns a list of indices (starting from 0) of the input columns that match the names/numbers given to cols. This is a low-level operation which is called by gal_table_read (described above), see there for more on each argument’s description. allcols is the returned array of gal_table_info.

void gal_table_comments_add_intro (gal_list_str_t **comments, char *program_string, time_t *rawtime)

Add some basic information to the list of comments. This basic information includes the following information

• If the program is run in a Git version controlled directory, Git’s description is printed (see description under COMMIT in Section 4.10 [Output FITS files], page 191).

• The calendar time that is stored in rawtime (time_t is C’s calendar time format defined in time.h). You can calculate the time in this format with the following expressions:

  time_t rawtime;
  time(&rawtime);

• The name of your program in program_string. If it is NULL, this line is ignored.
void gal_table_write (gal_data_t *cols, struct gal_fits_list_key_t **keywords, gal_list_str_t *comments, int tableformat, char *filename, char *extname, uint8_t colinfoinstdout)

Write cols (a list of datasets, see Section 11.3.8.9 [List of gal_data_t], page 566) into a table stored in filename. The format of the table can be determined with tableformat that accepts the macros defined above. When filename==NULL, the column information will be printed on the standard output (command-line).

If comments!=NULL, the list of comments (see Section 11.3.8.1 [List of strings], page 555) will also be printed into the output table. When the output table is a plain text file, every node of comments will be printed after a # (so it can be considered as a comment) and in FITS table they will follow a COMMENT keyword.

If a file named filename already exists, the operation depends on the type of output. When filename is a FITS file, the table will be added as a new extension after all existing extensions. If filename is a plain text file, this function will abort with an error.

If filename is a FITS file, the table extension will have the name extname.

When colinfoinstdout!=0 and filename==NULL (columns are printed in the standard output), the dataset metadata will also printed in the standard output. When printing to the standard output, the column information can be piped into another program for further processing and thus the meta-data (lines starting with a #) must be ignored. In such cases, you only print the column values by passing 0 to colinfoinstdout.

void gal_table_write_log (gal_data_t *logll, char *program_string, time_t *rawtime, gal_list_str_t *comments, char *filename, int quiet)

Write the logll list of datasets into a table in filename (see Section 11.3.8.9 [List of gal_data_t], page 566). This function is just a wrapper around gal_table_comments_add_intro and gal_table_write (see above). If quiet is non-zero, this function will print a message saying that the filename has been created.

11.3.11 FITS files (fits.h)

The FITS format is the most common format to store data (images and tables) in astronomy. The CFITSIO library already provides a very good low-level collection of functions for manipulating FITS data. The low-level nature of CFITSIO is defined for versatility and portability. As a result, even a simple and basic operation, like reading an image or table column into memory, will require a special sequence of CFITSIO function calls which can be inconvenient and buggy to manage in separate locations. To ease this process, Gnuastro's library provides wrappers for CFITSIO functions. With these, it much easier to read, write, or modify FITS file data, header keywords and extensions. Hence, if you feel these functions don’t exactly do what you want, we strongly recommend reading the CFITSIO manual to use its great features directly (afterwards, send us your wrappers so we can include it here for others to benefit also).

All the functions and macros introduced in this section are declared in gnuastro/fits.h. When you include this header, you are also including CFITSIO’s fitsio.h header. So you
don’t need to explicitly include fitsio.h anymore and can freely use any of its macros or functions in your code along with those discussed here.

### 11.3.11.1 FITS Macros, errors and filenames

Some general constructs provided by Gnuastro’s FITS handling functions are discussed here. In particular there are several useful functions about FITS file names.

**GAL_FITS_MAX_NDIM**

[Macro]

The maximum number of dimensions a dataset can have in FITS format, according to the FITS standard this is 999.

**void**

```
gal_fits_io_error (int status, char *message)
```

If `status` is non-zero, this function will print the CFITSIO error message corresponding to status, print `message` (optional) in the next line and abort the program. If `message==NULL`, it will print a default string after the CFITSIO error.

**int**

```
gal_fits_name_is_fits (char *name)
```

If the `name` is an acceptable CFITSIO FITS filename return 1 (one), otherwise return 0 (zero). The currently acceptable FITS suffixes are `.fits`, `.fit`, `.fits.gz`, `.fits.Z`, `.imh`, `.fits.fz`. IMH is the IRAF format which is acceptable to CFITSIO.

**int**

```
gal_fits_suffix_is_fits (char *suffix)
```

Similar to `gal_fits_name_is_fits`, but only for the suffix. The suffix doesn’t have to start with ‘.’: this function will return 1 (one) for both `fits` and `.fits`.

**in**

```
gal_fits_file_recognized (char *name)
```

Return 1 if the given file name (possibly including its contents) is a FITS file. This is necessary in when the contents of a FITS file do follow the FITS standard, but it the file doesn’t have a Gnuastro-recognized FITS suffix. Therefore, it will first call `gal_fits_name_is_fits`, if the result is negative, then this function will attempt to open the file with CFITSIO and if it works, it will close it again and return 1. In the process of opening the file, CFITSIO will just to open the file, no reading will take place, so it should have minimal CPU footprint.

**char **

```
gal_fits_name_save_as_string (char *filename, char *hdv)
```

If the name is a FITS name, then put a (hdv: ...) after it and return the string. If it isn’t a FITS file, just print the name, if `filename==NULL`, then return the string `stdin`. Note that the output string’s space is allocated.

This function is useful when you want to report a random file to the user which may be FITS or not (for a FITS file, simply the filename is not enough, the HDU is also necessary).
11.3.11.2 CFITSIO and Gnuastro types

Both Gnuastro and CFITSIO have special identifiers for each type that they accept. Gnuastro’s type identifiers are fully described in Section 11.3.3 [Library data types (type.h)], page 532, and are usable for all kinds of datasets (images, table columns, etc) as part of Gnuastro’s Section 11.3.6.1 [Generic data container (gal_data_t)], page 542. However, following the FITS standard, CFITSIO has different identifiers for images and tables. Following CFITSIO’s own convention, we will use bitpix for image type identifiers and datatype for its internal identifiers (and mainly used in tables). The functions introduced in this section can be used to convert between CFITSIO and Gnuastro’s type identifiers.

One important issue to consider is that CFITSIO’s types are not fixed width (for example long may be 32-bits or 64-bits on different systems). However, Gnuastro’s types are defined by their width. These functions will use information on the host system to do the proper conversion. To have a portable (usable on different systems) code, is thus recommended to use these functions and not to assume a fixed correspondence between CFITSIO and Gnuastro’s types.

```c
uint8_t gal_fits_bitpix_to_type (int bitpix)
    Return the Gnuastro type identifier that corresponds to CFITSIO’s bitpix on this system.

int gal_fits_type_to_bitpix (uint8_t type)
    Return the CFITSIO bitpix value that corresponds to Gnuastro’s type.

char gal_fits_type_to_bin_tform (uint8_t type)
    Return the FITS standard binary table TFORM character that corresponds to Gnuastro’s type.

int gal_fits_type_to_datatype (uint8_t type)
    Return the CFITSIO datatype that corresponds to Gnuastro’s type on this machine.

uint8_t gal_fits_datatype_to_type (int datatype, int is_table_column)
    Return Gnuastro’s type identifier that corresponds to the CFITSIO datatype. Note that when dealing with CFITSIO’s TLONG, the fixed width type differs between tables and images. So if the corresponding dataset is a table column, put a non-zero value into is_table_column.
```

11.3.11.3 FITS HDUs

A FITS file can contain multiple HDUs/extensions. The functions in this section can be used to get basic information about the extensions or open them. Note that fitsfile is defined in CFITSIO’s fitsio.h which is automatically included by Gnuastro’s gnuastro/fits.h.
fitsfile *

gal_fits_open_to_write (char *filename)
If filename exists, open it and return the fitsfile pointer that corresponds to it. If filename doesn’t exist, the file will be created which contains a blank first extension and the pointer to its next extension will be returned.

size_t

gal_fits_hdu_num (char *filename)
Return the number of HDUs/extensions in filename.

unsigned long

gal_fits_hdu_datasum (char *filename, char *hdu)
Return the DATASUM of the given HDU in the given FITS file. For more on DATASUM in the FITS standard, see Section 5.1.1.2 [Keyword inspection and manipulation], page 201, (under the checksum component of --write).

unsigned long

gal_fits_hdu_datasum_ptr (fitsfile *fptr)
Return the DATASUM of the already opened HDU in fptr. For more on DATASUM in the FITS standard, see Section 5.1.1.2 [Keyword inspection and manipulation], page 201, (under the checksum component of --write).

int

gal_fits_hdu_format (char *filename, char *hdu)
Return the format of the HDU as one of CFITSIO’s recognized macros: IMAGE_HDU, ASCII_TBL, or BINARY_TBL.

int

gal_fits_hdu_is_healpix (fitsfile *fptr)
Return 1 if the dataset may be a HEALpix grid and 0 otherwise. Technically, it is considered to be a HEALPix if the HDU isn’t an ASCII table, and has the NSIDE, FIRSTPIX and LASTPIX.

fitsfile *

gal_fits_hdu_open (char *filename, char *hdu, int iomode)
Open the HDU/extension hdu from filename and return a pointer to CFITSIO’s fitsfile. iomode determines how the FITS file will be opened using CFITSIO’s macros: READONLY or READWRITE.
The string in hdu will be appended to filename in square brackets so CFITSIO only opens this extension. You can use any formatting for the hdu that is acceptable to CFITSIO. See the description under --hdu in Section 4.1.2.1 [Input/Output options], page 157, for more.

fitsfile *

gal_fits_hdu_open_format (char *filename, char *hdu, int img0_tab1)
Open (in read-only format) the hdu HDU/extension of filename as an image or table. When img0_tab1 is 0(zero) but the HDU is a table, this function will abort with an error. It will also abort with an error when img0_tab1 is 1(one), but the HDU is an image.
A FITS HDU may contain both tables or images. When your program needs one of these formats, you can call this function so if the user provided the wrong HDU/file, it will abort and inform the user that the file/HDU is has the wrong format.

### 11.3.11.4 FITS header keywords

Each FITS extension/HDU contains a raw dataset which can either be a table or an image along with some header keywords. The keywords can be used to store meta-data about the actual dataset. The functions in this section describe Gnuastro’s high-level functions for reading and writing FITS keywords. Similar to all Gnuastro’s FITS-related functions, these functions are all wrappers for CFITSIO’s low-level functions.

The necessary meta-data (header keywords) for a particular dataset are commonly numerous, it is much more efficient to list them in one variable and call the reading/writing functions once. Hence the functions in this section use linked lists, a thorough introduction to them is given in Section 11.3.8 [Linked lists (list.h)], page 554. To reading FITS keywords, these functions use a list of Gnuastro’s generic dataset format that is discussed in Section 11.3.8.9 [List of gal_data_t], page 566. To write FITS keywords we define the gal_fits_list_key_t node that is defined below.

#### gal_fits_list_key_t

[Type (C struct)]

Structure for writing FITS keywords. This structure is used for one keyword and you don’t need to set all elements. With the next element, you can link it to another keyword thus creating a linked list to add any number of keywords easily and at any step during your program (see Section 11.3.8 [Linked lists (list.h)], page 554, for an introduction on lists). See the functions below for adding elements to the list.

```c
typedef struct gal_fits_list_key_t {
  int tfree; /* ==1, free title string. */
  int kfree; /* ==1, free keyword name. */
  int vfree; /* ==1, free keyword value. */
  int cfree; /* ==1, free comment. */
  int ufree; /* ==1, free unit. */
  uint8_t type; /* Keyword value type. */
  char *title; /* !NULL, only print title.*/
  char *keyname; /* Keyword Name. */
  void *value; /* Keyword value. */
  char *comment; /* Keyword comment. */
  char *unit; /* Keyword unit. */
  struct gal_fits_list_key_t *next; /* Pointer next keyword. */
} gal_fits_list_key_t;
```

#### int gal_fits_key_exists_fptr (fitsfile *fptr, char *keyname)

Return 1 (true) if the opened FITS file pointer contains the requested keyword and 0 (false) otherwise.
void * gal_fits_key_img_blank (uint8_t type)
Returns a pointer to an allocated space containing the value to the FITS BLANK header keyword, when the input array has a type of type. This is useful when you want to write the BLANK keyword using CFITSIO's fits_write_key function.

According to the FITS standard: “If the BSCALE and BZERO keywords do not have the default values of 1.0 and 0.0, respectively, then the value of the BLANK keyword must equal the actual value in the FITS data array that is used to represent an undefined pixel and not the corresponding physical value”. Therefore a special BLANK value is needed for datasets containing signed 8-bit, unsigned 16-bit, unsigned 32-bit, and unsigned 64-bit integers (types that are defined with BSCALE and BZERO in the FITS standard).

Not usable when reading a dataset: As quoted from the FITS standard above, the value returned by this function can only be generically used for the writing of the BLANK keyword header. It must not be used as the blank pointer when when reading a FITS array using CFITSIO. When reading an array with CFITSIO, you can use gal_blank_alloc_write to generate the necessary pointer.

void gal_fits_key_clean_str_value (char *string)
Remove the single quotes and possible extra spaces around the keyword values that CFITSIO returns when reading a string keyword. CFITSIO doesn’t remove the two single quotes around the string value of a keyword. Hence the strings it reads are like: 'value ', or 'some_very_long_value'. To use the value during your processing, it is commonly necessary to remove the single quotes (and possible extra spaces). This function will do this within the allocated space of the string.

char * gal_fits_key_date_to_struct_tm (char *fitsdate, struct tm *tp)
Parse fitsdate as a FITS date format string (most generally: YYYY-MM-DDThh:mm:ss.ddd...) into the C library’s broken-down time structure, or struct tm (declared in time.h) and return a pointer to a newly allocated array for the sub-second part of the format (.ddd...). Therefore it needs to be freed afterwards (if it isn’t NULL) When there is no sub-second portion, this pointer will be NULL.

This is a relatively low-level function, an easier function to use is gal_fits_key_date_to_seconds which will return the sub-seconds as double precision floating point.

Note that the FITS date format mentioned above is the most complete representation. The following two formats are also acceptable: YYYY-MM-DDThh:mm:ss and YYYY-MM-DD. This option can also interpret the older FITS date format where only two characters are given to the year and the date format is reversed (DD/MM/YYYYhh:mm:ssddd...). In this case (following the GNU C Library), this option will make the following assumption: values 68 to 99 correspond to the years 1969 to 1999, and values 0 to 68 as the years 2000 to 2068.
size_t gal_fits_key_date_to_seconds (char *fitsdate, char **subsecstr, double *subsec)

Return the Unix epoch time (number of seconds that have passed since 00:00:00 Thursday, January 1st, 1970) corresponding to the FITS date format string fitsdate (see description of gal_fits_key_date_to_struct_tm above). This function will return GAL_BLANK_SIZE_T if the broken-down time couldn’t be converted to seconds.

The Unix epoch time is in units of seconds, but the FITS date format allows sub-second accuracy. The last two arguments are for the optional sub-second portion. If you don’t want sub-second information, just set the second argument to NULL.

If fitsdate contains sub-second accuracy and subsecstr!=NULL, then the starting of the sub-second part’s string is stored in subsecstr (malloc’ed), and subsec will be the corresponding numerical value (between 0 and 1, in double precision floating point). So to avoid leaking memory, if a sub-second string is requested, it must be freed after calling this function. When a sub-second string doesn’t exist (and it is requested), then a value of NULL and NaN will be written in *subsecstr and *subsec respectively.

This is a very useful function for operations on the FITS date values, for example sorting FITS files by their dates, or finding the time difference between two FITS files. The advantage of working with the Unix epoch time is that you don’t have to worry about calendar details (for example the number of days in different months, or leap years, etc).

void gal_fits_key_read_from_ptr (fitsfile *fptr, gal_data_t *keysll, int readcomment, int readunit)

Read the list of keyword values from a FITS pointer. The input should be a linked list of Gnuastro’s generic data container (gal_data_t). Before calling this function, you just have to set the name, and optionally, the desired type of the value of each keyword. The given name value will be directly passed to CFITSIO to read the desired keyword name. This function will allocate space to keep the value. If no pre-defined type is requested for a certain keyword’s value, the smallest possible type to host the value will be found and used. If readcomment and readunit are non-zero, this function will also try to read the possible comments and units of the keyword.

Here is one example of using this function:

/* Allocate an array of datasets. */
gal_data_t *keysll=gal_data_array_calloc(N);

/* Make the array usable as a list too (by setting ‘next’). */
for(i=0;i<N-1;++i) keysll[i].next=&keysll[i+1];

/* Fill the datasets with a ‘name’ and a ‘type’. */
keysll[0].name="NAME1";    keysll[0].type=GAL_TYPE_INT32;
keysll[1].name="NAME2";    keysll[1].type=GAL_TYPE_STRING;
...
/* Call this function. */
gal_fits_key_read_from_ptr(fptr, keysll, 0, 0);

/* Use the values as you like... */

/* Free all the allocated spaces. Note that 'name' wasn't allocated in this example, so we should explicitly set it to NULL before calling 'gal_data_array_free'. */
for(i=0;i<N;++i) keysll[i].name=NULL;
gal_data_array_free(keysll, N, 1);

If the array pointer of each keyword’s dataset is not NULL, then it is assumed that the space to keep the value has already been allocated. If it is NULL, space will be allocated for the value by this function.

Strings need special consideration: the reason is that generally, gal_data_t needs to also allow for array of strings (as it supports arrays of integers for example). Hence when reading a string value, two allocations may be done by this function (one if array!=NULL).

Therefore, when using the values of strings after this function, keysll[i].array must be interpreted as char **: one allocation for the pointer, one for the actual characters. If you use something like the example, above you don’t have to worry about the freeing, gal_data_array_free will free both allocations. So to read a string, one easy way would be the following:

```c
char *str, **strarray;
strarr = keysll[i].array;
str    = strarray[0];
```

If CFITSIO is unable to read a keyword for any reason the status element of the respective gal_data_t will be non-zero. If it is zero, then the keyword was found and successfully read. Otherwise, it is a CFITSIO status value. You can use CFITSIO’s error reporting tools or gal_fits_io_error (see Section 11.3.11.1 [FITS Macros, errors and filenames], page 573) for reporting the reason of the failure. A tip: when the keyword doesn’t exist, CFITSIO’s status value will be KEY_NO_EXIST.

CFITSIO will start searching for the keywords from the last place in the header that it searched for a keyword. So it is much more efficient if the order that you ask for keywords is based on the order they are stored in the header.

```c
void [Function]
gal_fits_key_read (char *filename, char *hdu, gal_data_t *keysll, int readcomment, int readunit)
```

Same as gal_fits_read_keywords_fptr (see above), but accepts the filename and HDU as input instead of an already opened CFITSIO fitsfile pointer.
void gal_fits_key_list_add (gal.fits_list_key_t **list, uint8_t type, char *keyname, int kfree, void *value, int vfree, char *comment, int cfree, char *unit, int ufree)

Add a keyword to the top of list of header keywords that need to be written into a FITS file. In the end, the keywords will have to be freed, so it is important to know before hand if they were allocated or not (hence the presence of the arguments ending in free). If the space for the respective element is not allocated, set these arguments to 0 (zero).

You can call this function multiple times on a single list add several keys that will be written in one call to gal.fits_key_write or gal.fits_key_write_in_ptr. However, the resulting list will be a last-in-first-out list (for more on lists, see Section 11.3.8 [Linked lists (list.h)], page 554). Hence, the written keys will have the inverse order of your calls to this function. To avoid this problem, you can either use gal.fits_key_list_add_end instead (which will add each key to the end of the list, not to the top like this function). Alternatively, you can use gal.fits_key_list_reverse after adding all the keys with this function.

Important note for strings: the value should be the pointer to the string its-self (char *), not a pointer to a pointer (char **).

void gal_fits_key_list_add_end (gal.fits_list_key_t **list, uint8_t type, char *keyname, int kfree, void *value, int vfree, char *comment, int cfree, char *unit, int ufree)

Similar to gal.fits_key_list_add, but add the given keyword to the end of the list, see the description of gal.fits_key_list_add for more. Use this function if you want the keywords to be written in the same order that you add nodes to the list of keywords.

void gal_fits_key_list_title_add (gal.fits_list_key_t **list, char *title, int tfree)

Add a special “title” keyword (with the title string) to the top of the keywords list. If cfree is non-zero, the space allocated for comment will be freed immediately after writing the keyword (in another function).

void gal_fits_key_list_title_add_end (gal.fits_list_key_t **list, char *title, int tfree)

Similar to gal.fits_key_list_title_add, but put the comments at the end of the list.

void gal_fits_key_list_comment_add (gal.fits_list_key_t **list, char *comment, int ffree)

Add a COMMENT keyword to the top of the keywords list. If the comment is longer than 70 characters, CFITSIO will automatically break it into multiple COMMENT keywords.
If \texttt{fcfree} is non-zero, the space allocated for \texttt{comment} will be freed immediately after writing the keyword (in another function).

\begin{verbatim}
void gal_fits_key_list_comment_add_end (gal.fits_list_key_t **list, char *comment, int fcfree)
    Similar to \texttt{gal_fits_key_list_comment_add}, but put the comments at the end of the list.
\end{verbatim}

\begin{verbatim}
void gal_fits_key_list_reverse (gal.fits_list_key_t **list)
    Reverse the input list of keywords.
\end{verbatim}

\begin{verbatim}
void gal_fits_key_write_title_in_ptr (char *title, fitsfile *fptr)
    Add two lines of “title” keywords to the given CFITSIO \texttt{fptr} pointer. The first line will be blank and the second will have the string in \texttt{title} roughly in the middle of the line (a fixed distance from the start of the keyword line). A title in the list of keywords helps in classifying the keywords into groups and inspecting them by eye. If \texttt{title==NULL}, this function won’t do anything.
\end{verbatim}

\begin{verbatim}
void gal_fits_key_write_filename (char *keynamebase, char *filename, gal.fits_list_key_t **list, int top1end0, int quiet)
    Put \texttt{filename} into the \texttt{gal.fits_list_key_t} list (possibly broken up into multiple keywords) to later write into a HDU header. The \texttt{keynamebase} string will be appended with a \_\text{N} (N>0) and used as the keyword name. If \texttt{top1end0!=0}, then the keywords containing the filename will be added to the top of the list.

    The FITS standard sets a maximum length of 69 characters for the string values of a keyword\textsuperscript{20}. This creates problems with file names (which include directories) because file names/addresses can become longer than . Therefore, when \texttt{filename} is longer than the maximum length of a FITS keyword value, this function will break it into several keywords (breaking up the string on directory separators). So the full file/directory address (including directories) can be longer than 69 characters. However, if a single file or directory name (within a larger address) is longer than 69 characters, this function will truncate the name and print a warning. If \texttt{quiet!=0}, then the warning won’t be printed.
\end{verbatim}

\begin{verbatim}
void gal_fits_key_write_wcsstr (fitsfile *fptr, struct wcsprm wcs, char *wcsstr, int nkeyrec)
    Write the WCS header string (produced with WCSLIB’s \texttt{wcshdo} function) into the CFITSIO \texttt{fitsfile} pointer. \texttt{nkeyrec} is the number of FITS header keywords in \texttt{wcsstr}. This function will put a few blank keyword lines along with a comment \texttt{WCS information} before writing each keyword record.
\end{verbatim}

\textsuperscript{20} The limit is actually 71 characters (which is the full 80 character length, subtracted by 8 for the keyword name and one character for the =). However, for strings, FITS also requires two single quotes.
void gal_fits_key_write (gal.fits_list_key_t **keylist, char *title, char *filename, char *hdu)
  Write the list of keywords in keylist into the hdu extension of the file called filename (the file must already exist) and free the list.
  The list nodes are meant to be dynamically allocated (because they will be freed after being written). We thus recommend using the gal_fits_key_list_add or gal_fits_key_list_add_end to create and fill the list. Below is one fully working example of using this function to write a keyword into an existing FITS file.

  #include <stdio.h>
  #include <stdlib.h>
  #include <gnuastro/fits.h>

  int main()
  {
    char *filename="test.fits";
    gal.fits_list_key_t *keylist=NULL;
    char *unit="unit";
    float value=123.456;
    char *keyname="MYKEY";
    char *comment="A good description of the key";
    gal_fits_key_list_add_end(&keylist, GAL_TYPE_FLOAT32, keyname, 0, &value, 0, comment, 0, unit, 0);
    gal_fits_key_write(&keylist, "Matching metadata", filename, "1");
    return EXIT_SUCCESS;
  }

void gal_fits_key_write_in_ptr (gal.fits_list_key_t **keylist, fitsfile *fptr)
  Write the list of keywords in keylist into the given CFITSIO fitsfile pointer and free keylist. For more on the input keylist, see the description and example for gal_fits_key_write, above.

void gal_fits_key_write_version (gal.fits_list_key_t **keylist, char *title, char *filename, char *hdu)
  Write the (optional, when keylist!=NULL) given list of keywords under the optional FITS keyword title, then print all the important version and date information. This is basically, just a wrapper over gal_fits_key_write_version_in_ptr.

void gal_fits_key_write_version_in_ptr (gal.fits_list_key_t **keylist, char *title, fitsfile *fptr)
  Write or update (all the) keyword(s) in headers into the FITS pointer, but also the date, name of your program (program_name), along with the versions of CFITSIO, WCSLIB (when available), GSL, Gnuastro, and (the possible) commit information into the header as described in Section 4.10 [Output FITS files], page 191.
Since the data processing depends on the versions of the libraries you have used, it is strongly recommended to include this information in every FITS output. `gal.fits_img_write` and `gal.fits_tab_write` will automatically use this function.

```c
void gal_fits_key_write_config (gal.fits_list_key_t **keylist, char *title, char *extname, char *filename, char *hdu)
```

Write the given keyword list (`keylist`) into the `hdu` extension of `filename`, ending it with version information. This function will write `extname` as the name of the extension (value to the standard `EXTNAME` FITS keyword). The list of keywords will then be printed under a title called `title`.

This function is used by many Gnuastro programs and is primarily intended for writing configuration settings of a program into the zero-th extension of their FITS outputs (which is empty when the FITS file is created by Gnuastro's program and this library).

### 11.3.11.5 FITS arrays (images)

Images (or multi-dimensional arrays in general) are one of the common data formats that is stored in FITS files. Only one image may be stored in each FITS HDU/extension. The functions described here can be used to get the information of, read, or write images in FITS files.

```c
void gal_fits_img_info (fitsfile *fptr, int *type, size_t *ndim, size_t **dsize, char **name, char **unit)
```

Read the type (see Section 11.3.3 [Library data types (type.h)], page 532), number of dimensions, and size along each dimension of the CFITSIO `fitsfile` into the `type`, `ndim`, and `dsize` pointers respectively. If `name` and `unit` are not `NULL` (point to a `char *`), then if the image has a name and units, the respective string will be put in these pointers.

```c
size_t * gal_fits_img_info_dim (char *filename, char *hdu, size_t *ndim)
```

Put the number of dimensions in the `hdu` extension of `filename` in the space that `ndim` points to and return the size of the dataset along each dimension as an allocated array with `*ndim` elements.

```c
gal_data_t * gal_fits_img_read (char *filename, char *hdu, size_t minmapsize, int quietmmap)
```

Read the contents of the `hdu` extension/HDU of `filename` into a Gnuastro generic data container (see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542) and return it. If the necessary space is larger than `minmapsize`, then don’t keep the data in RAM, but in a file on the HDD/SSD. For more on `minmapsize` and `quietmmap` see the description under the same name in Section 11.3.6.1 [Generic data container (gal_data_t)], page 542.

Note that this function only reads the main data within the requested FITS extension, the WCS will not be read into the returned dataset. To read the WCS, you can use
The `gal_wcs_read` function as shown below. Afterwards, the `gal_data_free` function will free both the dataset and any WCS structure (if there are any).

```c
data=gal_fits_img_read(filename, hdu, -1, 1);
data->wcs=gal_wcs_read(filename, hdu, 0, 0, &data->wcs->nwcs);
```

**[Function]**

`gal_data_t * gal_fits_img_read_to_type (char *inputname, char *inhdu, uint8_t type, size_t minmmapsize, int quietmmap)`

Read the contents of the hdu extension/HDU of `filename` into a Gnuastro generic data container (see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542) of type `type` and return it.

This is just a wrapper around `gal_fits_img_read` (to read the image/array of any type) and `gal_data_copy_to_new_type_free` (to convert it to `type` and free the initially read dataset). See the description there for more.

**[Function]**

`gal_data_t * gal_fits_img_read_kernel (char *filename, char *hdu, size_t minmmapsize, int quietmmap)`

Read the hdu of `filename` as a convolution kernel. A convolution kernel must have an odd size along all dimensions, it must not have blank (NaN in floating point types) values and must be flipped around the center to make the proper convolution (see Section 6.3.1.1 [Convolution process], page 307). If there are blank values, this function will change the blank values to 0.0. If the input image doesn’t have the other two requirements, this function will abort with an error describing the condition to the user. The finally returned dataset will have a `float32` type.

**[Function]**

`fitsfile * gal_fits_img_write_to_ptr (gal_data_t *input, char *filename)`

Write the input dataset into a FITS file named `filename` and return the corresponding CFITSIO `fitsfile` pointer. This function won’t close `fitsfile`, so you can still add other extensions to it after this function or make other modifications.

**[Function]**

`void gal_fits_img_write (gal_data_t *data, char *filename, gal_fits_list_key_t *headers, char *program_string)`

Write the input dataset into the FITS file named `filename`. Also add the `headers` keywords to the newly created HDU/extension it along with your program’s name (`program_string`).

**[Function]**

`void gal_fits_img_write_to_type (gal_data_t *data, char *filename, gal_fits_list_key_t *headers, char *program_string, int type)`

Convert the input dataset into `type`, then write it into the FITS file named `filename`. Also add the `headers` keywords to the newly created HDU/extension along with your program’s name (`program_string`). After the FITS file is written, this function will free the copied dataset (with type `type`) from memory.

This is just a wrapper for the `gal_data_copy_to_new_type` and `gal_fits_img_write` functions.
void [Function]
gal_fits_img_write_corr_wcs_str (gal_data_t *data, char *filename, char
    *wcsstr, int nkeyrec, double *crpix, gal_fits_list_key_t *headers, char
    *program_string)
Write the input dataset into filename using the wcsstr while correcting the CRPIX
values.
This function is mainly useful when you want to make FITS files in parallel (from one
main WCS structure, with just differing CRPIX). This can happen in the following
cases for example:

- When a large number of FITS images (with WCS) need to be created in parallel,
  it can be much more efficient to write the header’s WCS keywords once at first,
  write them in the FITS file, then just correct the CRPIX values.
- WCSLIB’s header writing function is not thread safe. So when writing FITS
  images in parallel, we can’t write the header keywords in each thread.

11.3.11.6 FITS tables
Tables are one of the common formats of data that is stored in FITS files. Only one table
may be stored in each FITS HDU/extension, but each table column must be viewed as a
different dataset (with its own name, units and numeric data type for example). The only
constraint of the column datasets in a table is that they must be one-dimensional and have
the same number of elements as the other columns. The functions described here can be
used to get the information of, read, or write columns into FITS tables.

void [Function]
gal_fits_tab_size (fitsfile *fitsptr, size_t *nrows, size_t *ncols)
Read the number of rows and columns in the table within CFITSIO’s fitsptr.

int [Function]
gal_fits_tab_format (fitsfile *fitsptr)
Return the format of the FITS table contained in CFITSIO’s fitsptr. Recall that
FITS tables can be in binary or ASCII formats. This function will return GAL_TABLE-
FORMAT_AFITS or GAL_TABLE_FORMAT_BFITS (defined in Section 11.3.10 [Table input
output (table.h)], page 568). If the fitsptr is not a table, this function will abort
the program with an error message informing the user of the problem.

gal_data_t *
gal_fits_tab_info (char *filename, char *hdu, size_t *numcols, size_t
    *numrows, int *tableformat)
Store the information of each column in hdu of filename into an array of data structures
with numcols elements (one data structure for each column) see Section 11.3.6.3
[Arrays of datasets], page 548. The total number of rows in the table is also put into
the memory that numrows points to. The format of the table (e.g., FITS binary or
ASCII table) will be put in tableformat (macros defined in Section 11.3.10 [Table
input output (table.h)], page 568).
This function is just for column information. Therefore it only stores meta-data like
column name, units and comments. No actual data (contents of the columns for
example the array or dsize elements) will be allocated by this function. This is
Chapter 11: Library

a low-level function particular to reading tables in FITS format. To be generic, it is recommended to use `gal_table_info` which will allow getting information from a variety of table formats based on the filename (see Section 11.3.10 [Table input output (table.h)], page 568).

```c
[Function]
gal_data_t *
gal_fits_tab_read (char *filename, char *hdu, size_t numrows, gal_data_t *colinfo, gal_list_sizet_t *indexll, size_t numthreads, size_t minmapsize, int quietmmap)
```

Read the columns given in the list `indexll` from a FITS table (in `filename` and HDU/extension `hdu`) into the returned linked list of data structures, see Section 11.3.8.3 [List of `size_t`], page 558, and Section 11.3.8.9 [List of `gal_data_t`], page 566.

Each column will be read independently, therefore they will be read in `numthreads` CPU threads to greatly speed up the reading when there are many columns and rows. However, this only happens if CFITSIO was configured with `--enable-reentrant`. This test has been done at Gnuastro’s configuration time; if so, `GAL_CONFIG_HAVE_FITS_IS_REENTRANT` will have a value of 1, otherwise, it will have a value of 0.

For more on this macro, see Section 11.3.1 [Configuration information (config.h)], page 526.

If the necessary space for each column is larger than `minmapsize`, don’t keep it in the RAM, but in a file in the HDD/SSD. For more on `minmapsize` and `quietmmap`, see the description under the same name in Section 11.3.6.1 [Generic data container (gal_data_t)], page 542.

Each column will have `numrows` rows and `colinfo` contains any further information about the columns (returned by `gal_fits_tab_info`, described above). Note that this is a low-level function, so the output data linked list is the inverse of the input indexes linked list. It is recommended to use `gal_table_read` for generic reading of tables, see Section 11.3.10 [Table input output (table.h)], page 568.

```c
[Function]
void
gal_fits_tab_write (gal_data_t *cols, gal_list_str_t *comments, int tableformat, char *filename, char *extname)
```

Write the list of datasets in `cols` (see Section 11.3.8.9 [List of `gal_data_t`], page 566) as separate columns in a FITS table in `filename`. If `filename` already exists then this function will write the table as a new extension called `extname`, after all existing ones. The format of the table (ASCII or binary) may be specified with the `tableformat` (see Section 11.3.10 [Table input output (table.h)], page 568). If `comments!=NULL`, each node of the list of strings will be written as a COMMENT keywords in the output FITS file (see Section 11.3.8.1 [List of strings], page 555).

This is a low-level function for tables. It is recommended to use `gal_table_write` for generic writing of tables in a variety of formats, see Section 11.3.10 [Table input output (table.h)], page 568.

11.3.12 File input output

The most commonly used file format in astronomical data analysis is the FITS format (see Section 5.1 [Fits], page 194, for an introduction), therefore Gnuastro’s library pro-
vides a large and separate collection of functions to read/write data from/to them (see Section 11.3.11 [FITS files (fits.h)], page 572). However, FITS is not well recognized outside the astronomical community and cannot be imported into documents or slides. Therefore, in this section, we discuss the other different file formats that Gnuastro's library recognizes.

11.3.12.1 Text files (txt.h)

The most universal and portable format for data storage are plain text files. They can be viewed and edited on any text editor or even on the command-line. This section describes some functions that help in reading from and writing to plain text files.

Lines are one of the most basic building blocks (delimiters) of a text file. Some operating systems like Microsoft Windows, terminate their ASCII text lines with a carriage return character and a new-line character (two characters, also known as CRLF line terminators). While Unix-like operating systems just use a single new-line character. The functions below that read an ASCII text file are able to identify lines with both kinds of line terminators.

Gnuastro defines a simple format for metadata of table columns in a plain text file that is discussed in Section 4.7.2 [Gnuastro text table format], page 185. The functions to get information from, read from and write to plain text files also follow those conventions.

**Macros**

- `GAL_TXT_LINESTAT_INVALID` [Macro]
- `GAL_TXT_LINESTAT_BLANK` [Macro]
- `GAL_TXT_LINESTAT_COMMENT` [Macro]
- `GAL_TXT_LINESTAT_DATAROW` [Macro]

Status codes for lines in a plain text file that are returned by `gal_txt_line_stat`. Lines which have a # character as their first non-white character are considered to be comments. Lines with nothing but white space characters are considered blank. The remaining lines are considered as containing data.

**Function**

- `int gal_txt_line_stat (char *line)`
  
  Check the contents of `line` and see if it is a blank, comment, or data line. The returned values are the macros that start with `GAL_TXT_LINESTAT`.

- `char * gal_txt_trim_space (char *str)`
  
  Trim the white space characters before and after the given string. The operation is done within the allocated space of the string, so if you need the string untouched, please pass an allocated copy of the string to this function. The returned pointer is within the input string. If the input pointer is NULL, or the string only has white-space characters, the returned pointer will be NULL.

- `gal_data_t * gal_txt_table_info (char *filename, gal_list_str_t *lines, size_t *numcols, size_t *numrows)`
  
  Store the information of each column in a text file `filename`, or list of strings (`lines`) into an array of data structures with `numcols` elements (one data structure for each column) see Section 11.3.6.3 [Arrays of datasets], page 548. The total number of rows in the table is also put into the memory that `numrows` points to.
lines is a list of strings with each node representing one line (including the new-line character), see Section 11.3.8.1 [List of strings], page 555. It will mostly be the output of gal_txt_stdin_read, which is used to read the program’s input as separate lines from the standard input (see below). Note that filename and lines are mutually exclusive and one of them must be NULL.

This function is just for column information. Therefore it only stores meta-data like column name, units and comments. No actual data (contents of the columns for example the array or dsize elements) will be allocated by this function. This is a low-level function particular to reading tables in plain text format. To be generic, it is recommended to use gal_table_info which will allow getting information from a variety of table formats based on the filename (see Section 11.3.10 [Table input output (table.h)], page 568).

```
[Function]
gal_data_t *
gal_txt_table_read (char *filename, gal_list_str_t *lines, size_t numrows,
                   gal_data_t *colinfo, gal_list_sizet_t *indexll, size_t minmapsize, int quietmmap)
```

Read the columns given in the list indexll from a plain text file (filename) or list of strings (lines), into a linked list of data structures (see Section 11.3.8.3 [List of size_t], page 558, and Section 11.3.8.9 [List of gal_data_t], page 566). If the necessary space for each column is larger than minmapsize, don’t keep it in the RAM, but in a file on the HDD/SSD. For more one minmapsize and quietmmap, see the description under the same name in Section 11.3.6.1 [Generic data container (gal_data_t)], page 542.

lines is a list of strings with each node representing one line (including the new-line character), see Section 11.3.8.1 [List of strings], page 555. It will mostly be the output of gal_txt_stdin_read, which is used to read the program’s input as separate lines from the standard input (see below). Note that filename and lines are mutually exclusive and one of them must be NULL.

Note that this is a low-level function, so the output data list is the inverse of the input indices linked list. It is recommended to use gal_table_read for generic reading of tables in any format, see Section 11.3.10 [Table input output (table.h)], page 568.

```
[Function]
gal_data_t *
gal_txt_image_read (char *filename, gal_list_str_t *lines, size_t minmapsize,
                    int quietmmap)
```

Read the 2D plain text dataset in file (filename) or list of strings (lines) into a dataset and return the dataset. If the necessary space for the image is larger than minmapsize, don’t keep it in the RAM, but in a file on the HDD/SSD. For more on minmapsize and quietmmap, see the description under the same name in Section 11.3.6.1 [Generic data container (gal_data_t)], page 542.

lines is a list of strings with each node representing one line (including the new-line character), see Section 11.3.8.1 [List of strings], page 555. It will mostly be the output of gal_txt_stdin_read, which is used to read the program’s input as separate lines from the standard input (see below). Note that filename and lines are mutually exclusive and one of them must be NULL.
gal_list_str_t * gal_txt_stdin_read (long timeout_microsec)

Read the complete standard input and return a list of strings with each line (including the new-line character) as one node of that list. If the standard input is already filled (for example connected to another program’s output with a pipe), then this function will parse the whole stream.

If Standard input is not pre-configured and the first line is typed/written in the terminal before timeout_microsec micro-seconds, it will continue parsing until reaches an end-of-file character (CTRL-D after a new-line on the keyboard) with no time limit. If nothing is entered before timeout_microsec micro-seconds, it will return NULL. All the functions that can read plain text tables will accept a filename as well as a list of strings (intended to be the output of this function for using Standard input). The reason for keeping the standard input is that once something is read from the standard input, it is hard to put it back. We often need to read a text file several times: once to count how many columns it has and which ones are requested, and another time to read the desired columns. So it easier to keep it all in allocated memory and pass it on from the start for each round.

void gal_txt_write (gal_data_t *cols, struct gal.fits_list_key_t **keylist,
                    gal_list_str_t *comment, char *filename, uint8_t colinfoinstdout)

Write cols in a plain text file filename. cols may have one or two dimensions which determines the output:

1D  cols is treated as a column and a list of datasets (see Section 11.3.8.9 [List of gal_data_t], page 566): every node in the list is written as one column in a table.

2D  cols is a two dimensional array, it cannot be treated as a list (only one 2D array can currently be written to a text file). So if cols->next!=NULL the next nodes in the list are ignored and will not be written.

This is a low-level function for tables. It is recommended to use gal_table_write for generic writing of tables in a variety of formats, see Section 11.3.10 [Table input output (table.h)], page 568.

It is possible to add two types of metadata to the printed table: comments and keywords. Each string in the list given to comments will be printed into the file as a separate line, starting with #. Keywords have a more specific and computer-parsable format and are passed through keylist. Each keyword is also printed in one line, but with the format below. Because of the various components in a keyword, it is thus necessary to use the gal.fits_list_key_t data structure. For more, see Section 11.3.11.4 [FITS header keywords], page 576.

# [key] NAME: VALUE / [UNIT] KEYWORD COMMENT.

If filename already exists this function will abort with an error and will not write over the existing file. Before calling this function make sure if the file exists or not. If comments!=NULL, a # will be put at the start of each node of the list of strings and will be written in the file before the column meta-data in filename (see Section 11.3.8.1 [List of strings], page 555).
When `filename==NULL`, the column information will be printed on the standard output (command-line). When `colinfoinstdout!=0` and `filename==NULL` (columns are printed in the standard output), the dataset metadata will also printed in the standard output. When printing to the standard output, the column information can be piped into another program for further processing and thus the meta-data (lines starting with a #) must be ignored. In such cases, you only print the column values by passing 0 to `colinfoinstdout`.

11.3.12.2 TIFF files (tiff.h)

Outside of astronomy, the TIFF standard is arguably the most commonly used format to store high-precision data/images. Unlike FITS however, the TIFF standard only supports images (not tables), but like FITS, it has support for all standard data types (see Section 4.5 [Numeric data types], page 178) which is the primary reason other fields use it.

Another similarity of the TIFF and FITS standards is that TIFF supports multiple images in one file. The TIFF standard calls each one of these images (and their accompanying meta-data) a ‘directory’ (roughly equivalent to the FITS extensions). Unlike FITS however, the directories can only be identified by their number (counting from zero), recall that in FITS you can also use the extension name to identify it.

The functions described here allow easy reading (and later writing) of TIFF files within Gnuastro or for users of Gnuastro’s libraries. Currently only reading is supported, but if you are interested, please get in touch with us.

```c
int gal_tiff_name_is_tiff (char *name)

Return 1 if `name` has a TIFF suffix. This can be used to make sure that a given input file is TIFF. See `gal_tiff_suffix_is_tiff` for a list of recognized suffixes.
```

```c
int gal_tiff_suffix_is_tiff (char *name)

Return 1 if `suffix` is a recognized TIFF suffix. The recognized suffixes are `tif`, `tiff`, `TIFF` and `TIFF`.
```

```c
size_t gal_tiff_dir_string_read (char *string)

Return the number within `string` as a `size_t` number to identify a TIFF directory. Note that the directories start counting from zero.
```

```c
gal_data_t * gal_tiff_read (char *filename, size_t dir, size_t minmmapsize, int quietmmap)

Read the `dir` directory within the TIFF file `filename` and return the contents of that TIFF directory as `gal_data_t`. If the directory’s image contains multiple channels, the output will be a list (see Section 11.3.8.9 [List of `gal_data_t`], page 566).
```

11.3.12.3 JPEG files (jpeg.h)

The JPEG file format is one of the most common formats for storing and transferring images, recognized by almost all image rendering and processing programs. In particular, because of its lossy compression algorithm, JPEG files can have low volumes, making it
used heavily on the internet. For more on this file format, and a comparison with others, please see Section 5.2.1 [Recognized file formats], page 211.

For scientific purposes, the lossy compression and very limited dynamic range (8-bit integers) make JPEG very unattractive for storing of valuable data. However, because of its commonality, it will inevitably be needed in some situations. The functions here can be used to read and write JPEG images into Gnuastro’s Section 11.3.6.1 [Generic data container (\texttt{gal\_data\_t})], page 542. If the JPEG file has more than one color channel, each channel is treated as a separate node in a list of datasets (see Section 11.3.8.9 [List of \texttt{gal\_data\_t}], page 566).

\begin{verbatim}
int gal_jpeg_name_is_jpeg (char *name)
    Return 1 if \texttt{name} has a JPEG suffix. This can be used to make sure that a given input file is JPEG. See \texttt{gal\_jpeg\_suffix\_is\_jpeg} for a list of recognized suffixes.

int gal_jpeg_suffix_is_jpeg (char *name)
    Return 1 if \texttt{suffix} is a recognized JPEG suffix. The recognized suffixes are \texttt{.jpg}, \texttt{.JPg}, \texttt{.jpeg}, \texttt{.JPEG}, \texttt{.jpe}, \texttt{.jif}, \texttt{.jfif} and \texttt{.jfi}.

gal_data_t *gal_jpeg_read (char *filename, size_t minmapsize, int quietmmap)
    Read the JPEG file \texttt{filename} and return the contents as \texttt{gal\_data\_t}. If the directory’s image contains multiple colors/channels, the output will be a list with one node per color/channel (see Section 11.3.8.9 [List of \texttt{gal\_data\_t}], page 566).

void gal_jpeg_write (gal_data_t *in, char *filename, uint8_t quality, float widthincm)
    Write the given dataset (\texttt{in}) into \texttt{filename} (a JPEG file). If \texttt{in} is a list, then each node in the list will be a color channel, therefore there can only be 1, 3 or 4 nodes in the list. If the number of nodes is different, then this function will abort the program with a message describing the cause. The lossy JPEG compression level can be set through \texttt{quality} which is a value between 0 and 100 (inclusive, 100 being the best quality). The display width of the JPEG file in units of centimeters (to suggest to viewers/users, only a meta-data) can be set through \texttt{widthincm}.
\end{verbatim}

11.3.12.4 EPS files (\texttt{eps.h})

The Encapsulated PostScript (EPS) format is commonly used to store images (or individual/single-page parts of a document) in the PostScript documents. For a more complete introduction, please see Section 5.2.1 [Recognized file formats], page 211. To provide high quality graphics, the Postscript language is a vectorized format, therefore pixels (elements of a “rasterized” format) aren’t defined in their context.

To display rasterized images, PostScript does allow arrays of pixels. However, since the over-all EPS file may contain many vectorized elements (for example borders, text, or other lines over the text) and interpreting them is not trivial or necessary within Gnuastro’s scope, Gnuastro only provides some functions to write a dataset (in the \texttt{gal\_data\_t} format, see Section 11.3.6.1 [Generic data container (\texttt{gal\_data\_t})], page 542) into EPS.
Chapter 11: Library

int [Function]
gal_eps_name_is_eps (char *name)
    Return 1 if name has an EPS suffix. This can be used to make sure that a given input file is EPS. See gal_eps_suffix_is_eps for a list of recognized suffixes.

int [Function]
gal_eps_suffix_is_eps (char *name)
    Return 1 if suffix is a recognized EPS suffix. The recognized suffixes are .eps, .EPS, .epsf, .epsi.

void [Function]
gal_eps_to_pt (float widthincm, size_t *dsize, size_t *w_h_in_pt)
    Given a specific width in centimeters (widthincm and the number of the dataset's pixels in each dimension (dsize) calculate the size of the output in PostScript points. The output values are written in the w_h_in_pt array (which has to be allocated before calling this function). The first element in w_h_in_pt is the width and the second is the height of the image.

void [Function]
gal_eps_write (gal_data_t *in, char *filename, float widthincm, uint32_t borderwidth, int hex, int dontoptimize, int forpdf)
    Write the in dataset into an EPS file called filename. in has to be an unsigned 8-bit character type (GAL_TYPE_UINT8, see Section 4.5 [Numeric data types], page 178). The desired width of the image in human/non-pixel units (to help the displayer) can be set with the widthincm argument. If borderwidth is non-zero, it is interpreted as the width (in points) of a solid black border around the image. A border can helpful when importing the EPS file into a document.

    EPS files are plain-text (can be opened/edited in a text editor), therefore there are different encodings to store the data (pixel values) within them. Gnuastro supports the Hexadecimal and ASCII85 encoding. ASCII85 is more efficient (producing small file sizes), so it is the default encoding. To use Hexadecimal encoding, set hex to a non-zero value. Currently If you don’t directly want to import the EPS file into a PostScript document but want to later compile it into a PDF file, set the forpdf argument to 1.

    By default, when the dataset only has two values, this function will use the PostScript optimization that allows setting the pixel values per bit, not byte (Section 5.2.1 [Recognized file formats], page 211). This can greatly help reduce the file size. However, when dontoptimize!=0, this optimization is disabled: even though there are only two values (is binary), the difference between them does not correspond to the full contrast of black and white.

11.3.12.5 PDF files (pdf.h)

The portable document format (PDF) has arguably become the most common format used for distribution of documents. In practice, a PDF file is just a compiled PostScript file. For a more complete introduction, please see Section 5.2.1 [Recognized file formats], page 211. To provide high quality graphics, the PDF is a vectorized format, therefore pixels (elements of a “rasterized” format) aren’t defined in their context. As a result, similar to Section 11.3.12.4 [EPS files (eps.h)], page 591, Gnuastro only writes datasets to a PDF file, not vice-versa.
int gal_pdf_name_is_pdf (char *name)

Return 1 if name has a PDF suffix. This can be used to make sure that a given input file is PDF. See gal_pdf_suffix_is_pdf for a list of recognized suffixes.

int gal_pdf_suffix_is_pdf (char *name)

Return 1 if suffix is a recognized PDF suffix. The recognized suffixes are .pdf and .PDF.

void gal_pdf_write (gal_data_t *in, char *filename, float widthincm, uint32_t borderwidth, int dontoptimize)

Write the in dataset into an EPS file called filename. in has to be an unsigned 8-bit character type (GAL_TYPE_UINT8, see Section 4.5 [Numeric data types], page 178). The desired width of the image in human/non-pixel units (to help the displayer) can be set with the widthincm argument. If borderwidth is non-zero, it is interpreted as the width (in points) of a solid black border around the image. A border can helpful when importing the PDF file into a document.

This function is just a wrapper for the gal_eps_write function in Section 11.3.12.4 [EPS files (eps.h)], page 591. After making the EPS file, Ghostscript (with a version of 9.10 or above, see Section 3.1.2 [Optional dependencies], page 122) will be used to compile the EPS file to a PDF file. Therefore if GhostScript doesn’t exist, doesn’t have the proper version, or fails for any other reason, the EPS file will remain. It can be used to find the cause, or use another converter or PostScript compiler.

By default, when the dataset only has two values, this function will use the PostScript optimization that allows setting the pixel values per bit, not byte (Section 5.2.1 [Recognized file formats], page 211). This can greatly help reduce the file size. However, when dontoptimize!=0, this optimization is disabled: even though there are only two values (is binary), the difference between them does not correspond to the full contrast of black and white.

11.3.13 World Coordinate System (wcs.h)

The FITS standard defines the world coordinate system (WCS) as a mechanism to associate physical values to positions within a dataset. For example, it can be used to convert pixel coordinates in an image to celestial coordinates like the right ascension and declination. The functions in this section are mainly just wrappers over CFITSIO, WCSLIB and GSL library functions to help in common applications.

GAL_WCS_DISTORTION_TPD [Macro]
GAL_WCS_DISTORTION_SIP [Macro]
GAL_WCS_DISTORTION_TPV [Macro]
GAL_WCS_DISTORTION_DSS [Macro]
GAL_WCS_DISTORTION_WAT [Macro]
Chapter 11: Library

GAL_WCS_DISTORTION_INVALID

Gnuastro identifiers of the various WCS distortion conventions, for more, see Calabretta et al. (2004, preprint)\textsuperscript{21}. Among these, SIP is a prior distortion, the rest other are sequent distortions. TPD is a superset of all these, hence it has both prior and sequent distortion coefficients. More information is given in the documentation of dis.h, from the WCSLIB manual\textsuperscript{22}.

GAL_WCS_COORDSYS_EQB1950
GAL_WCS_COORDSYS_EQJ2000
GAL_WCS_COORDSYS_ECB1950
GAL_WCS_COORDSYS_ECJ2000
GAL_WCS_COORDSYS_GALACTIC
GAL_WCS_COORDSYS_SUPERGALACTIC
GAL_WCS_COORDSYS_INVALID


GAL_WCS_LINEAR_MATRIX_PC
GAL_WCS_LINEAR_MATRIX_CD
GAL_WCS_LINEAR_MATRIX_INVALID

Identifiers of the linear transformation matrix: either in the $\text{PC}_i\_j$ or the $\text{CD}_i\_j$ formalism. For more, see the description of \texttt{--wcslinearmatrix} in Section 4.1.2.1 [Input/Output options], page 157.

GAL_WCS_FLTERROR

Limit of rounding for floating point errors.

\textbf{struct wcsprm *} \texttt{gal_wcs_read_fitsptr} (fitsfile *fptr, int linearmatrix, size_t hstartwcs, size_t hendwcs, int *nwcs)

[Not thread-safe] Return the WCSLIB wcsprm structure that is read from the CFITS-SIO fptr pointer to an opened FITS file. Also put the number of coordinate representations found into the space that nwcs points to. To read the WCS structure directly from a filename, see \texttt{gal_wcs_read} below. After processing has finished, you can free the returned structure with WCSLIB's \texttt{wcsfree} keyword:

\begin{verbatim}
    status = wcsfree(&nwcs,&wcs);
\end{verbatim}

The linearmatrix argument takes one of three values: 0, GAL\_WCS\_LINEAR\_MATRIX\_PC and GAL\_WCS\_LINEAR\_MATRIX\_CD. It will determine the format of the WCS when it is later written to file with \texttt{gal_wcs_write} or \texttt{gal_wcs_write_in_fitsptr} (which is called by \texttt{gal.fits_img_write}) So if you don’t want to write the WCS into a file later, just give it a value of 0. For more on the difference between these modes, see the description of \texttt{--wcslinearmatrix} in Section 4.1.2.1 [Input/Output options], page 157.

\textsuperscript{21} \url{https://www.atnf.csiro.au/people/mcalabre/WCS/dcs_20040422.pdf}
\textsuperscript{22} \url{https://www.atnf.csiro.au/people/mcalabre/WCS/wcslib/dis_8h.html}
If you don’t want to search the full FITS header for WCS-related FITS keywords (for example due to conflicting keywords), but only a specific range of the header keywords you can use the `hstartwcs` and `hendwcs` arguments to specify the keyword number range (counting from zero). If `hendwcs` is larger than `hstartwcs`, then only keywords in the given range will be checked. Hence, to ignore this feature (and search the full FITS header), give both these arguments the same value.

If the WCS information couldn’t be read from the FITS file, this function will return a NULL pointer and put a zero in `nwcs`. A WCSLIB error message will also be printed in `stderr` if there was an error.

This function is just a wrapper over WCSLIB’s `wcspih` function which is not thread-safe. Therefore, be sure to not call this function simultaneously (over multiple threads).

```c
struct wcsprm * gal_wcs_read (char *filename, char *hdu, int linear_matrix, size_t hstartwcs, size_t hendwcs, int *nwcs)
```

[Not thread-safe] Return the WCSLIB structure that is read from the HDU/extension `hdu` of the file `filename`. Also put the number of coordinate representations found into the space that `nwcs` points to. Please see `gal_wcs_read_fitsptr` for more.

```c
struct wcsprm * gal_wcs_create (double *crpix, double *crval, double *cdelt, double *pc, char **cunit, char **ctype, size_t ndim, int linear_matrix)
```

Given all the most common standard components of the WCS standard, construct a `struct wcsprm`, initialize and set it for future processing. See the FITS WCS standard for more on these keywords. All the arrays must have `ndim` elements with them except for `pc` which should have `ndim*ndim` elements (a square matrix). Also, `cunit` and `ctype` are arrays of strings. If `GAL_WCS_LINEAR_MATRIX_CD` is passed to `linear_matrix` then the output WCS structure will have a CD matrix (even though you have given a PC and CDELT matrix as input to this function). Otherwise, the output will have a PC and CDELT matrix (which is the recommended format by WCSLIB).

```c
char * gal_wcs_dimension_name (struct wcsprm *wcs, size_t dimension)
```

Return an allocated string array (that should be freed later) containing the first part of the `CTYPEi` FITS keyword (which contains the dimension name in the FITS standard). For example if `CTYPE1` is `RA---TAN`, the string that function returns will be `RA`. Recall that the second component of `CTYPEi` contains the type of projection.

```c
char * gal_wcs_write_wcsstr (struct wcsprm *wcs, int *nkeyrec)
```

Return an allocated string which contains the respective FITS keywords for the given WCS structure into it. The number of keywords is written in the space pointed by `nkeyrec`. Each FITS keyword is 80 characters wide (according to the FITS standard), and the next one is placed immediately after it, so the full string has `80*nkeyrec` bytes. The output of this function can later be written into an opened FITS file
using `gal_fits_key_writewcsstr` (see Section 11.3.11.4 [FITS header keywords], page 576).

```c
void

void

gal_wcs_remove_dimension(struct wcsprm *wcs, size_t fitsdim)
Remove the given FITS dimension from the given wcs structure.

void

gal_wcs_on_tile(gal_data_t *tile)
Create a WCSLIB `wcsprm` structure for `tile` using WCS parameters of the tile's allocated block dataset, see Section 11.3.15 [Tessellation library (tile.h)], page 608, for the definition of tiles. If `tile` already has a WCS structure, this function won't do anything.

In many cases, tiles are created for internal/low-level processing. Hence for performance reasons, when creating the tiles they don't have any WCS structure. When needed, this function can be used to add a WCS structure to each tile tile by copying the WCS structure of its block and correcting the reference point's coordinates within the tile.

double *

gal_wcs_warp_matrix(struct wcsprm *wcs)
Return the Warping matrix of the given WCS structure as an array of double precision floating points. This will be the final matrix, irrespective of the type of storage in the WCS structure. Recall that the FITS standard has several methods to store the matrix. The output is an allocated square matrix with each side equal to the number of dimensions.
void

**gal_wcs_clean_small_errors** (struct wcsprm *wcs)

Errors can make small differences between the pixel-scale elements (CDELT) and can also lead to extremely small values in the PC matrix. With this function, such errors will be “cleaned” as follows: 1) if the maximum difference between the CDELT elements is smaller than the reference error, it will be set to the mean value. When the FITS keyword CRDER (optional) is defined it will be used as a reference, if not the default value is GAL_WCS_FLTERROR. 2) If any of the PC elements differ from 0, 1 or -1 by less than GAL_WCS_FLTERROR, they will be rounded to the respective value.

void

**gal_wcs_decompose_pc_cdel** (struct wcsprm *wcs)

Decompose the PCi_j and CDELTi elements of wcs. According to the FITS standard, in the PCi_j WCS formalism, the rotation matrix elements m_{ij} are encoded in the PCi_j keywords and the scale factors are encoded in the CDELTi keywords. There is also another formalism (the CDi_j formalism) which merges the two into one matrix. However, WCSLIB’s internal operations are apparently done in the PCi_j formalism. So its outputs are also all in that format by default. When the input is a CDi_j, WCSLIB will still read the matrix directly into the PCi_j matrix and the CDELTi values are set to 1 (one). This function is designed to correct such issues: after it is finished, the CDELTi values in wcs will correspond to the pixel scale, and the PCi_j will correction show the rotation.

void

**gal_wcs_to_cd** (struct wcsprm *wcs)

Make sure that the WCS structure’s PCi_j and CDi_j keywords have the same value and that the CDELTi keywords have a value of 1.0. Also, set the wcs->altlin=2 (for the CDi_j formalism). With these changes gal_wcs_write_in_fitsptr (and thus gal_wcs_write and gal_fits_img_write and its derivatives) will have an output file in the format of CDi_j.

int

**gal_wcs_coordsys_from_string** (char *coordsys)

Convert the given string to Gnuastro’s integer-based WCS coordinate system identifier (one of the GAL_WCS_COORDSYS_* listed above). The expected strings can be seen in the description of the --wcscoordsys option of the Fits program, see Section 5.1.1.2 [Keyword inspection and manipulation], page 201.

int

**gal_wcs_coordsys_identify** (struct wcsprm *wcs)

Read the given WCS structure and return its coordinate system as one of Gnuastro’s WCS coordinate system identifiers (the macros GAL_WCS_COORDSYS_* listed above).

struct wcsprm *

**gal_wcs_coordsys_convert** (struct wcsprm *inwcs, int coordsysid)

Return a newly allocated WCS structure with the coordsysid coordinate system identifier. The Gnuastro WCS distortion identifiers are defined in the GAL_WCS_COORDSYS_* macros mentioned above. Since the returned dataset is newly allocated,
if you don’t need the original dataset after this, use the WCSLIB library function `wcsfree` to free the input, for example `wcsfree(inwcs)`.

```c
int gal_wcs_distortion_from_string (char *distortion)
Convert the given string (assumed to be a FITS-standard, string-based distortion identifier) to a Gnuastro's integer-based distortion identifier (one of the GAL_WCS_DISTORTION_* macros defined above). The sting-based distortion identifiers have three characters and are all in capital letters.
```

```c
int gal_wcs_distortion_to_string (int distortion)
Convert the given Gnuastro integer-based distortion identifier (one of the GAL_WCS_DISTORTION_* macros defined above) to the string-based distortion identifier) of the FITS standard. The sting-based distortion identifiers have three characters and are all in capital letters.
```

```c
int gal_wcs_distortion_identify (struct wcsprm *wcs)
Returns the Gnuastro identifier for the distortion of the input WCS structure. The returned value is one of the GAL_WCS_DISTORTION_* macros defined above. When the input pointer to a structure is NULL, or it doesn’t contain a distortion, the returned value will be GAL_WCS_DISTORTION_INVALID.
```

```c
struct wcsprm *
gal_wcs_distortion_convert(struct wcsprm *inwcs, int outdisptype, size_t *fitsize)
Return a newly allocated WCS structure, where the distortion is implemented in a different standard, identified by the identifier outdisptype. The Gnuastro WCS distortion identifiers are defined in the GAL_WCS_DISTORTION_* macros mentioned above.

The available conversions in this function will grow. Currently it only supports converting TPV to SIP and vice versa, following the recipe of Shupe et al. (2012)\textsuperscript{23}. Please get in touch with us if you need other types of conversions.

For some conversions, direct analytical conversions don’t exist. It is thus necessary to model and fit the two types. In such cases, it is also necessary to specify the fitsize array that is the size of the array along each C-ordered dimension, so you can simply pass the dsize element of your gal_data_t dataset, see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542. Currently this is only necessary when converting TPV to SIP. For other conversions you may simply pass a NULL pointer.

For example, if you want to convert the TPV coefficients of your input `image.fits` to SIP coefficients, you can use the following functions (which are also available as a command-line operation in Section 5.1 [Fits], page 194).

```c
int nwcs;
gal_data_t *data=gal_fits_img_read("image.fits", "1", -1, 1);
```

```c
inwcs = gal_wcs_read("image.fits", "1", 0, &nwcs);
data->wcs = gal_wcs_distortion_convert(inwcs, GAL_WCS_DISTORTION_TPV, NULL);
wcsfree(inwcs);
gal_fits_img_write(data, "tpv.fits", NULL, NULL);
```

### Function

**`double gal_wcs_angular_distance_deg (double r1, double d1, double r2, double d2)`**

Return the angular distance (in degrees) between a point located at \((r_1, d_1)\) to \((r_2, d_2)\). All input coordinates are in degrees. The distance (along a great circle) on a sphere between two points is calculated with the equation below.

\[
\cos(d) = \sin(d_1) \sin(d_2) + \cos(d_1) \cos(d_2) \cos(r_1 - r_2)
\]

However, since the pixel scales are usually very small numbers, this function won’t use that direct formula. It will be use the Haversine formula (https://en.wikipedia.org/wiki/Haversine_formula) which is better considering floating point errors:

\[
\frac{\sin^2(d)}{2} = \sin^2 \left( \frac{d_1 - d_2}{2} \right) + \cos(d_1) \cos(d_2) \sin^2 \left( \frac{r_1 - r_2}{2} \right)
\]

**Function**

**`double * gal_wcs_pixel_scale (struct wcsprm *wcs)`**

Return the pixel scale for each dimension of \(wcs\) in degrees. The output is an allocated array of double precision floating point type with one element for each dimension. If its not successful, this function will return NULL.

**Function**

**`double gal_wcs_pixel_area_arcsec2 (struct wcsprm *wcs)`**

Return the pixel area of \(wcs\) in arc-second squared. This only works when the input dataset has at least two dimensions and the units of the first two dimensions (CUNIT keywords) are deg (for degrees). In other cases, this function will return a NaN.

**Function**

**`int gal_wcs_coverage (char *filename, char *hdu, size_t *ondim, double **ocenter, double **owidth, double **omin, double **omax)`**

Find the sky coverage of the image HDU (hdu) within filename. The number of dimensions is written into ndim, and space for the various output arrays is internally allocated and filled with the respective values.

**Function**

**`gal_data_t * gal_wcs_world_to_img (gal_data_t *coords, struct wcsprm *wcs, int inplace)`**

Convert the linked list of world coordinates in coords to a linked list of image coordinates given the input WCS structure. coords must be a linked list of data structures of float64 (‘double’) type, seeSection 11.3.8 [Linked lists (list.h)], page 554, and Section 11.3.8.9 [List of gal_data_t], page 566. The top (first popped/read) node
of the linked list must be the first WCS coordinate (RA in an image usually) etc. Similarly, the top node of the output will be the first image coordinate (in the FITS standard).

If `inplace` is zero, then the output will be a newly allocated list and the input list will be untouched. However, if `inplace` is non-zero, the output values will be written into the input’s already allocated array and the returned pointer will be the same pointer to `coords` (in other words, you can ignore the returned value). Note that in the latter case, only the values will be changed, things like units or name (if present) will be untouched.

```
[Function]
gal_data_t *
gal_wcs_img_to_world (gal_data_t *coords, struct wcsprm *wcs, int inplace)
```

Convert the linked list of image coordinates in `coords` to a linked list of world coordinates given the input WCS structure. See the description of `gal_wcs_world_to_img` for more details.

### 11.3.14 Arithmetic on datasets (arithmetic.h)

When the dataset’s type and other information are already known, any programming language (including C) provides some very good tools for various operations (including arithmetic operations like addition) on the dataset with a simple loop. However, as an author of a program, making assumptions about the type of data, its dimensions and other basic characteristics will come with a large processing burden.

For example if you always read your data as double precision floating points for a simple operation like addition with an integer constant, you will be wasting a lot of CPU and memory when the input dataset is `int32` type for example (see Section 4.5 [Numeric data types], page 178). This overhead may be small for small images, but as you scale your process up and work with hundred/thousands of files that can be very large, this overhead will take a significant portion of the processing power. The functions and macros in this section are designed precisely for this purpose: to allow you to do any of the defined operations on any dataset with no overhead (in the native type of the dataset).

Gnuastro’s Arithmetic program uses the functions and macros of this section, so please also have a look at the Section 6.2 [Arithmetic], page 272, program and in particular Section 6.2.3 [Arithmetic operators], page 276, for a better description of the operators discussed here.

The main function of this library is `gal_arithmetic` that is described below. It can take an arbitrary number of arguments as operands (depending on the operator, similar to `printf`). Its first two arguments are integers specifying the flags and operator. So first we will review the constants for the recognized flags and operators and discuss them, then introduce the actual function.

```
GAL_ARITHMETIC_FLAG_INPLACE [Macro]
GAL_ARITHMETIC_FLAG_FREE [Macro]
GAL_ARITHMETIC_FLAG_NUMOK [Macro]
GAL_ARITHMETIC_FLAG_ENVSEED [Macro]
GAL_ARITHMETIC_FLAG_QUIET [Macro]
```
**GAL\_ARITHMETIC\_FLAGS\_BASIC**

Bit-wise flags to pass onto `gal_arithmetic` (see below). To pass multiple flags, use the bitwise-or operator. For example if you pass `GAL\_ARITHMETIC\_FLAG\_INPLACE | GAL\_ARITHMETIC\_FLAG\_NUMOK`, then the operation will be done in-place (without allocating a new array), and a single number will also be acceptable (that will be applied to all the pixels). Each flag is described below:

**GAL\_ARITHMETIC\_FLAG\_INPLACE**

Do the operation in-place (in the input dataset, thus modifying it) to improve CPU and memory usage. If this flag is used, after `gal_arithmetic` finishes, the input dataset will be modified. It is thus useful if you have no more need for the input after the operation.

**GAL\_ARITHMETIC\_FLAG\_FREE**

Free (all the) input dataset(s) after the operation is done. Hence the inputs are no longer usable after `gal_arithmetic`.

**GAL\_ARITHMETIC\_FLAG\_NUMOK**

It is acceptable to use a number and an array together. For example if you want to add all the pixels in an image with a single number you can pass this flag to avoid having to allocate a constant array the size of the image (with all the pixels having the same number).

**GAL\_ARITHMETIC\_FLAG\_ENVSEED**

Use the pre-defined environment variable for setting the random number generator seed when an operator needs it (for example `mknoise-sigma`). For more on random number generation in Gnuastro see Section 8.2.1.4 [Generating random numbers], page 464.

**GAL\_ARITHMETIC\_FLAG\_QUIET**

Don’t print any warnings or messages for operators that may benefit from it. For example by default the `mknoise-sigma` operator prints the random number generator function and seed that it used (in case the user wants to reproduce this result later). By activating this bit flag to the call, that extra information is not printed on the command-line.

**GAL\_ARITHMETIC\_FLAGS\_BASIC**

A wrapper for activating the three “basic” operations that are commonly necessary together: `GAL\_ARITHMETIC\_FLAG\_INPLACE`, `GAL\_ARITHMETIC\_FLAG\_FREE` and `GAL\_ARITHMETIC\_FLAG\_NUMOK`.

**GAL\_ARITHMETIC\_OP\_PLUS**

[Macro]

**GAL\_ARITHMETIC\_OP\_MINUS**

[Macro]

**GAL\_ARITHMETIC\_OP\_MULTIPLY**

[Macro]

**GAL\_ARITHMETIC\_OP\_DIVIDE**

[Macro]

**GAL\_ARITHMETIC\_OP\_LT**

[Macro]

**GAL\_ARITHMETIC\_OP\_LE**

[Macro]

**GAL\_ARITHMETIC\_OP\_GT**

[Macro]

**GAL\_ARITHMETIC\_OP\_GE**

[Macro]

**GAL\_ARITHMETIC\_OP\_EQ**

[Macro]

**GAL\_ARITHMETIC\_OP\_NE**

[Macro]
GAL_ARITHMETIC_OP_AND  [Macro]
GAL_ARITHMETIC_OP_OR    [Macro]

Binary operators (requiring two operands) that accept datasets of any recognized type (see Section 4.5 [Numeric data types], page 178). When \texttt{gal\_arithmetic} is called with any of these operators, it expects two datasets as arguments. For a full description of these operators with the same name, see Section 6.2.3 [Arithmetic operators], page 276. The first dataset/operand will be put on the left of the operator and the second will be put on the right. The output type of the first four is determined from the input types (largest type of the inputs). The rest (which are all conditional operators) will output a binary uint8\_t (or unsigned char) dataset with values of either 0 (zero) or 1 (one).

GAL_ARITHMETIC_OP_NOT   [Macro]

The logical NOT operator. When \texttt{gal\_arithmetic} is called with this operator, it only expects one operand (dataset), since this is a unary operator. The output is uint8\_t (or unsigned char) dataset of the same size as the input. Any non-zero element in the input will be 0 (zero) in the output and any 0 (zero) will have a value of 1 (one).

GAL_ARITHMETIC_OP_ISBLANK  [Macro]

A unary operator with output that is 1 for any element in the input that is blank, and 0 for any non-blank element. When \texttt{gal\_arithmetic} is called with this operator, it will only expect one input dataset. The output dataset will have uint8\_t (or unsigned char) type.

\texttt{gal\_arithmetic} with this operator is just a wrapper for the \texttt{gal\_blank\_flag} function of Section 11.3.5 [Library blank values (\texttt{blank.h})], page 538, and this operator is just included for completeness in arithmetic operations. So in your program, it might be easier to just call \texttt{gal\_blank\_flag}.

GAL_ARITHMETIC_OP_WHERE   [Macro]

The three-operand \texttt{where} operator thoroughly discussed in Section 6.2.3 [Arithmetic operators], page 276. When \texttt{gal\_arithmetic} is called with this operator, it will only expect three input datasets: the first (which is the same as the returned dataset) is the array that will be modified. The second is the condition dataset (that must have a uint8\_t or unsigned char type), and the third is the value to be used if condition is non-zero.

As a result, note that the order of operands when calling \texttt{gal\_arithmetic} with \texttt{GAL\_ARITHMETIC\_OP\_WHERE} is the opposite of running Gnuastro’s Arithmetic program with the \texttt{where} operator (see Section 6.2 [Arithmetic], page 272). This is because the latter uses the reverse-Polish notation which isn’t necessary when calling a function (see Section 6.2.1 [Reverse polish notation], page 272).

GAL_ARITHMETIC_OP_SQRT   [Macro]
GAL_ARITHMETIC_OP_LOG     [Macro]
GAL_ARITHMETIC_OP_LOG10   [Macro]

Unary operator functions for calculating the square root (√i), ln(i) and log(i) mathematical operators on each element of the input dataset. The returned dataset will
have a floating point type, but its precision is determined from the input: if the input is a 64-bit floating point, the output will also be 64-bit. Otherwise, the returned dataset will be 32-bit floating point: you don’t gain precision by using these operators, but you gain in operating speed if you use the sufficient precision. See Section 4.5 [Numeric data types], page 178, for more on the precision of floating point numbers to help in selecting your required floating point precision.

If you want your output to be 64-bit floating point but your input is a different type, you can convert the input to a 64-bit floating point type with `gal_data_copy_to_new_type` or `gal_data_copy_to_new_type_free`, (see Section 11.3.6.4 [Copying datasets], page 549). Alternatively, you can use the `GAL_ARITHMETIC_OP_TO_FLOAT64` operators in the arithmetic library.

```
GAL_ARITHMETIC_OP_SIN [Macro]
GAL_ARITHMETIC_OP_COS [Macro]
GAL_ARITHMETIC_OP_TAN [Macro]
GAL_ARITHMETIC_OP_ASIN [Macro]
GAL_ARITHMETIC_OP_ACOS [Macro]
GAL_ARITHMETIC_OP_ATAN [Macro]
GAL_ARITHMETIC_OP_ATAN2 [Macro]
```

Trigonometric functions (and their inverse). All the angles, either inputs or outputs, are in units of degrees.

```
GAL_ARITHMETIC_OP_SINH [Macro]
GAL_ARITHMETIC_OP_COSH [Macro]
GAL_ARITHMETIC_OP_TANH [Macro]
GAL_ARITHMETIC_OP_ASINH [Macro]
GAL_ARITHMETIC_OP_ACOSH [Macro]
GAL_ARITHMETIC_OP_ATANH [Macro]
```

Hyperbolic functions (and their inverse).

```
GAL_ARITHMETIC_OP_RA_TO_DEGREE [Macro]
GAL_ARITHMETIC_OP_DEC_TO_DEGREE [Macro]
GAL_ARITHMETIC_OP_DEGREE_TO_RA [Macro]
GAL_ARITHMETIC_OP_DEGREE_TO_DEC [Macro]
```

Unary operators to convert between degrees (as a single floating point number) to the sexagesimal Right Ascension and Declination format (as strings, respectively in the format of \_h\_m\_s and \_d\_m\_s). The first two operators expect a string operand (in the sexagesimal formats mentioned above, but also in the \_:\_:\_) and will return a double-precision floating point operand. The latter two are the opposite.

```
GAL_ARITHMETIC_OP_MINVAL [Macro]
GAL_ARITHMETIC_OP_MAXVAL [Macro]
GAL_ARITHMETIC_OP_NUMBERVAL [Macro]
GAL_ARITHMETIC_OP_SUMVAL [Macro]
GAL_ARITHMETIC_OP_MEANVAL [Macro]
GAL_ARITHMETIC_OP_STDVAL [Macro]
GAL_ARITHMETIC_OP_MEDIANVAL [Macro]
```

Unary operand statistical operators that will return a single value for datasets of any size. These are just wrappers around similar functions in Section 11.3.22 [Statistical
operations (statistics.h), page 633, and are included in gal_arithmetic only for completeness (to use easily in Section 6.2 [Arithmetic], page 272). In your programs, it will probably be easier if you use those gal_statistics_ functions directly.

GAL_ARITHMETIC_OP_ABS [Macro]
Unary operand absolute-value operator.

GAL_ARITHMETIC_OP_MIN [Macro]
GAL_ARITHMETIC_OP_MAX [Macro]
GAL_ARITHMETIC_OP_NUMBER [Macro]
GAL_ARITHMETIC_OP_SUM [Macro]
GAL_ARITHMETIC_OP_MEAN [Macro]
GAL_ARITHMETIC_OP_STD [Macro]
GAL_ARITHMETIC_OP_MEDIAN [Macro]
Multi-operand statistical operations. When gal_arithmetic is called with any of these operators, it will expect only a single operand that will be interpreted as a list of datasets (see Section 11.3.8.9 [List of gal_data_t], page 566). These operators can work on multiple threads using the numthreads argument. See the discussion under the min operator in Section 6.2.3 [Arithmetic operators], page 276.

The output will be a single dataset with each of its elements replaced by the respective statistical operation on the whole list. The type of the output is determined from the operator (irrespective of the input type): for GAL_ARITHMETIC_OP_MIN and GAL_ARITHMETIC_OP_MAX, it will be the same type as the input, for GAL_ARITHMETIC_OP_NUMBER, the output will be GAL_TYPE_UINT32 and for the rest, it will be GAL_TYPE_FLOAT32.

GAL_ARITHMETIC_OP_QUANTILE [Macro]
Similar to the operands above (including GAL_ARITHMETIC_MIN), except that when gal_arithmetic is called with these operators, it requires two arguments. The first is the list of datasets like before, and the second is the 1-element dataset with the quantile value. The output type is the same as the inputs.

GAL_ARITHMETIC_OP_SIGCLIP_STD [Macro]
GAL_ARITHMETIC_OP_SIGCLIP_MEAN [Macro]
GAL_ARITHMETIC_OP_SIGCLIP_MEDIAN [Macro]
GAL_ARITHMETIC_OP_SIGCLIP_NUMBER [Macro]
Similar to the operands above (including GAL_ARITHMETIC_MIN), except that when gal_arithmetic is called with these operators, it requires two arguments. The first is the list of datasets like before, and the second is the 2-element list of $\sigma$-clipping parameters. The first element in the parameters list is the multiple of sigma and the second is the termination criteria (see Section 7.1.3 [Sigma clipping], page 343). The output type of GAL_ARITHMETIC_OP_SIGCLIP_NUMBER will be GAL_TYPE_UINT32 and for the rest it will be GAL_TYPE_FLOAT32.

GAL_ARITHMETIC_OP_MKNOISE_SIGMA [Macro]
GAL_ARITHMETIC_OP_MKNOISE_POISSON [Macro]
GAL_ARITHMETIC_OP_MKNOISE_UNIFORM [Macro]
Add noise to the input dataset. These operators take two arguments: the first is the input data set (can have any dimensionality or number of elements. The second
argument is the noise specifier (a single element, of any type): for a fixed-sigma noise, it is the Gaussian standard deviation, for the Poisson noise, it is the background (see Section 8.2.1.1 [Photon counting noise], page 462) and for the uniform distribution it is the width of the interval around each element of the input dataset.

By default, a separate random number generator seed will be used on each separate run of these operators. Therefore two identical runs on the same input will produce different results. You can get reproducible results by setting the GAL_RNG_SEED environment variable and activating the GAL_ARITHMETIC_FLAG_ENVSEED flag. For more on random number generation in Gnuastro, see Section 8.2.1.4 [Generating random numbers], page 464.

By default these operators will print the random number generator function and seed (in case the user wants to reproduce the result later), but this can be disabled by activating the bit-flag GAL_ARITHMETIC_FLAG_QUIET described above.

**GAL_ARITHMETIC_OP_SIZE**

Size operator that will return a single value for datasets of any kind. When gal_arithmetic is called with this operator, it requires two arguments. The first is the dataset, and the second is a single integer value. The output type is a single integer.

**GAL_ARITHMETIC_OP_POW**

Binary operator to-power operator. When gal_arithmetic is called with any of these operators, it will expect two operands: raising the first by the second (returning a floating point, inputs can be integers).

**GAL_ARITHMETIC_OP_BITAND**  
**GAL_ARITHMETIC_OP_BITOR**  
**GAL_ARITHMETIC_OP_BITXOR**  
**GAL_ARITHMETIC_OP_BITLSH**  
**GAL_ARITHMETIC_OP_BITRSH**  
**GAL_ARITHMETIC_OP_MODULO**

Binary integer-only operand operators. These operators are only defined on integer data types. When gal_arithmetic is called with any of these operators, it will expect two operands: the first is put on the left of the operator and the second on the right. The ones starting with BIT are the respective bit-wise operators in C and MODULO is the modulo/remainder operator. For a discussion on these operators, please see Section 6.2.3 [Arithmetic operators], page 276.

The output type is determined from the input types and C’s internal conversions: it is strongly recommended that both inputs have the same type (any integer type), otherwise the bit-wise behavior will be determined by your compiler.

**GAL_ARITHMETIC_OP_BITNOT**

The unary bit-wise NOT operator. When gal_arithmetic is called with any of these operators, it will expect one operand of an integer type and perform the bitwise-NOT operation on it. The output will have the same type as the input.

**GAL_ARITHMETIC_OP_TO_UINT8**  
**GAL_ARITHMETIC_OP_TO_INT8**  
**GAL_ARITHMETIC_OP_TO_UINT16**
Unary type-conversion operators. When \texttt{gal\_arithmetic} is called with any of these operators, it will expect one operand and convert it to the requested type. Note that with these operators, \texttt{gal\_arithmetic} is just a wrapper over the \texttt{gal\_data\_copy\_to\_new\_type} or \texttt{gal\_data\_copy\_to\_new\_type\_free} that are discussed in Copying datasets. It accepts these operators only for completeness and easy usage in Section 6.2 [Arithmetic], page 272. So in your programs, it might be preferable to directly use those functions.

\texttt{GAL\_ARITHMETIC\_OP\_BOX\_AROUND\_ELLIPSE}

Return the width (along horizontal) and height (along vertical) of a box that encompasses an ellipse with the same center point. For more on the three input operands to this operator see the description of box-around-ellipse. This function returns two datasets as a \texttt{gal\_data\_t} linked list. The top element of the list is the height and its next element is the width.

\texttt{GAL\_ARITHMETIC\_OP\_MAKENEW}

Create a new, zero-valued dataset with an unsigned 8-bit data type. The length along each dimension of the dataset should be given as a single list of \texttt{gal\_data\_t} \texttt{t}. The number of dimensions is derived from the number of nodes in the list and the length along each dimension is the single-valued element within that list. Just note that the list should be in the reverse of the desired dimensions.

\texttt{gal\_data\_t *}

\texttt{gal\_arithmetic (int operator, size\_t numthreads, int flags, ...)}

Apply the requested arithmetic operator on the operand(s). The \texttt{operator} is identified through the macros above (that start with \texttt{GAL\_ARITHMETIC\_OP\_}). The number of necessary operands (number of arguments to replace ‘...’ in the declaration of this function, above) depends on the operator and is described under each operator, above. Each operand has a type of \texttt{gal\_data\_t *} (see last paragraph with example).

If the operator can work on multiple threads, the number of threads can be specified with \texttt{numthreads}. When the operator is single-threaded, \texttt{numthreads} will be ignored. Special conditions can also be specified with the \texttt{flag} operator (a bit-flag with bits described above, for example \texttt{GAL\_ARITHMETIC\_FLAG\_INPLACE} or \texttt{GAL\_ARITHMETIC\_FLAG\_FREE}).

\texttt{gal\_arithmetic} is a multi-argument function (like C’s \texttt{printf}). In other words, the number of necessary arguments is not fixed and depends on the value to \texttt{operator}. Below, you can see a minimal, fully working example, showing how different operators need different numbers of arguments.

```c
#include <stdio.h>
#include <stdlib.h>
```
#include <gnuastro/fits.h>
#include <gnuastro/arithmetic.h>

int main(void)
{
    /* Define the datasets and flag. */
    gal_data_t *in1, *in2, *out1, *out2;
    int flag=GAL_ARITHMETIC_FLAGS_BASIC;

    /* Read the input images. */
    in1=gal_fits_img_read("image1.fits", "1", -1, 1);
    in2=gal_fits_img_read("image2.fits", "1", -1, 1);

    /* Take the logarithm (base-e) of the first input. */
    out1=gal_arithmetic(GAL_ARITHMETIC_OP_LOG, 1, flag, in1);

    /* Add the second input with the logarithm of the first. */
    out2=gal_arithmetic(GAL_ARITHMETIC_OP_PLUS, 1, flag, in2, out1);

    /* Write the output into a file. */
    gal_fits_img_write(out2, "out.fits", NULL, NULL);

    /* Clean up. Due to the in-place flag (in */
    /* 'GAL_ARITHMETIC_FLAGS_BASIC'), 'out1' and 'out2' point to the */
    /* same array in memory and due to the freeing flag, any input */
    /* dataset(s) that weren't returned have been freed internally by */
    /* 'gal_arithmetic'. Therefore it is only necessary to free */
    /* 'out2': all other allocated spaces have been freed internally. */
    /* before reaching this point. */
    gal_data_free(out2);

    /* Return control back to the OS (saying that we succeeded). */
    return EXIT_SUCCESS;
}

As you see above, you can feed the returned dataset from one call of `gal_arithmetic` to another call. The advantage of using `gal_arithmetic` (as opposed to manually writing a `for` or `while` loop and doing the operation with the `+` operator and `log()` function yourself), is that you don’t have to worry about the type of the input data (for a list of acceptable data types in Gnuastro, see Section 11.3.3 [Library data types (`type.h`)], page 532). Arithmetic will automatically deal with the data types internally and choose the best output type depending on the operator.

```
int gal_arithmetic_set_operator (char *string, size_t *num_operands)
    Return the operator macro/code that corresponds to `string`. The number of
    operands that it needs are written into the space that `*num_operands` points
to. If the string couldn’t be interpreted as an operator, this function will return
GAL_ARITHMETIC_OP_INVALID.

This function will check string with the fixed human-readable names (using strcmp)
for the operators and return the two numbers. Note that string must only contain
the single operator name and nothing else (not even any extra white space).

char *
gal_arithmetic_operator_string (int operator)

    Return the human-readable standard string that corresponds to the given operator.
    For example when the input is GAL_ARITHMETIC_OP_PLUS or GAL_ARITHMETIC_OP_-MEAN, the strings + or mean will be returned.

11.3.15 Tessellation library (tile.h)

In many contexts, it is desirable to slice the dataset into subsets or tiles (overlapping or
not). In such a way that you can work on each tile independently. One method would be
to copy that region to a separate allocated space, but in many contexts this isn’t necessary
and in fact can be a big burden on CPU/Memory usage. The block pointer in Gnuastro’s
Section 11.3.6.1 [Generic data container (gal_data_t)], page 542, is defined for such sit-
uations: where allocation is not necessary. You just want to read the data or write to it
independently (or in coordination with) other regions of the dataset. Added with parallel
processing, this can greatly improve the time/memory consumption.

See the figure below for example: assume the larger dataset is a contiguous block of
memory that you are interpreting as a 2D array. But you only want to work on the smaller
tile region.

```
+---------------------------------+
|   |   |
|   | tile |
|   |   |
|   |   |
| tile->block = larger |
+---------------------------------+
```

To use gal_data_t’s block concept, you allocate a gal_data_t *tile which is initial-
ized with the pointer to the first element in the sub-array (as its array argument). Note
that this is not necessarily the first element in the larger array. You can set the size of the
tile along with the initialization as you please. Recall that, when given a non-NULL pointer
as array, gal_data_initialize (and thus gal_data_alloc) do not allocate any space
and just uses the given pointer for the new array element of the gal_data_t. So your tile
data structure will not be pointing to a separately allocated space.

After the allocation is done, you just point tile->block to the larger dataset which
hosts the full block of memory. Where relevant, Gnuastro’s library functions will check the
block pointer of their input dataset to see how to deal with dimensions and increments so they can always remain within the tile. The tools introduced in this section are designed to help in defining and working with tiles that are created in this manner.

Since the block structure is defined as a pointer, arbitrary levels of tessellation/grid-ing are possible (tile->block may itself be a tile in an even larger allocated space). Therefore, just like a linked-list (see Section 11.3.8 [Linked lists (list.h)], page 554), it is important to have the block pointer of the largest (allocated) dataset set to NULL. Normally, you won’t have to worry about this, because gal_data_initialize (and thus gal_data_alloc) will set the block element to NULL by default, just remember not to change it. You can then only change the block element for the tiles you define over the allocated space.

Below, we will first review constructs for Section 11.3.15.1 [Independent tiles], page 609, and then define the current approach to fully tessellating a dataset (or covering every pixel/data-element with a non-overlapping tile grid in Section 11.3.15.2 [Tile grid], page 614. This approach to dealing with parts of a larger block was inspired from a similarly named concept in the GNU Scientific Library (GSL), see its “Vectors and Matrices” chapter for their implementation.

11.3.15.1 Independent tiles

The most general application of tiles is to treat each independently, for example they may overlap, or they may not cover the full image. This section provides functions to help in checking/inspecting such tiles. In Section 11.3.15.2 [Tile grid], page 614, we will discuss functions that define/work-with a tile grid (where the tiles don’t overlap and fully cover the input dataset). Therefore, the functions in this section are general and can be used for the tiles produced by that section also.

void gal_tile_start_coord (gal_data_t *tile, size_t *start_coord)  
[Function]  
Calculate the starting coordinates of a tile in its allocated block of memory and write them in the memory that start_coord points to (which must have tile->ndim elements).

void gal_tile_start_end_coord (gal_data_t *tile, size_t *start_end, int rel_block)  
[Function]  
Put the starting and ending (end point is not inclusive) coordinates of tile into the start_end array. It is assumed that a space of 2*tile->ndim has been already allocated (static or dynamic) for start_end before this function is called.

rel_block (or relative-to-block) is only relevant when tile has an intermediate tile between it and the allocated space (like a channel, see gal_tile_full_two_layers). If it doesn’t (tile->block points the allocated dataset), then the value to rel_block is irrelevant.

When tile->block is its self a larger block and rel_block is set to 0, then the starting and ending positions will be based on the position within tile->block, not the allocated space.
void * \[Function\]
gal_tile_start_end_ind_inclusive (gal_data_t *tile, gal_data_t *work, size_t *start_end_inc)

Put the indices of the first/start and last/end pixels (inclusive) in a tile into the start_end array (that must have two elements). NOTE: this function stores the index of each point, not its coordinates. It will then return the pointer to the start of the tile in the work data structure (which doesn’t have to be equal to tile->block).

The outputs of this function are defined to make it easy to parse over an n-dimensional tile. For example, this function is one of the most important parts of the internal processing of in GAL_TILE_PARSE_OPERATE function-like macro that is described below.

gal_data_t * \[Function\]
gal_tile_series_from_minmax (gal_data_t *block, size_t *minmax, size_t number)

Construct a list of tile(s) given coordinates of the minimum and maximum of each tile. The minimum and maximums are assumed to be inclusive and in C order (slowest dimension first). The returned pointer is an allocated gal_data_t array that can later be freed with gal_data_array_free (see Section 11.3.6.3 [Arrays of datasets], page 548). Internally, each element of the output array points to the next element, so the output may also be treated as a list of datasets (see Section 11.3.8.9 [List of gal_data_t], page 566) and passed onto the other functions described in this section.

The array keeping the minimum and maximum coordinates for each tile must have the following format. So in total minmax must have 2*ndim*number elements.

| min0_d0 | min0_d1 | max0_d0 | max0_d1 | ... |
| ... | minN_d0 | minN_d1 | maxN_d0 | maxN_d1 |

gal_data_t * \[Function\]
gal_tile_block (gal_data_t *tile)

Return the dataset that contains tile’s allocated block of memory. If tile is immediately defined as part of the allocated block, then this is equivalent to tile->block. However, it is possible to have multiple layers of tiles (where tile->block is itself a tile). So this function is the most generic way to get to the actual allocated dataset.

size_t \[Function\]
gal_tile_block_increment (gal_data_t *block, size_t *tsize, size_t num_increment, size_t *coord)

Return the increment necessary to start at the next contiguous patch memory associated with a tile. block is the allocated block of memory and tsize is the size of the tile along every dimension. If coord is NULL, it is ignored. Otherwise, it will contain the coordinate of the start of the next contiguous patch of memory.

This function is intended to be used in a loop and num_increment is the main variable to this function. For the first time you call this function, it should be 1. In subsequent calls (while you are parsing a tile), it should be increased by one.
gal_data_t * gal_tile_block_write_const_value (gal_data_t *tilevalues, gal_data_t *tilesll, int withblank, int initialize)
Write a constant value for each tile over the area it covers in an allocated dataset that is the size of tile's allocated block of memory (found through gal_tile_block described above). The arguments to this function are:

tilevalues This must be an array that has the same number of elements as the nodes in in tilesll and in the same order that 'tilesll' elements are parsed (from top to bottom, see Section 11.3.8 [Linked lists (list.h)], page 554). As a result the array's number of dimensions is irrelevant, it will be parsed contiguously.

tilesll The list of input tiles (see Section 11.3.8.9 [List of gal_data_t], page 566). Internally, it might be stored as an array (for example the output of gal_tile_series_from_minmax described above), but this function doesn't care, it will parse the next elements to go to the next tile. This function will not pop-from or free the tilesll, it will only parse it from start to end.

withblank If the block containing the tiles has blank elements, those blank elements will be blank in the output of this function also, hence the array will be initialized with blank values when this option is called (see below).

initialize Initialize the allocated space with blank values before writing in the constant values. This can be useful when the tiles don’t cover the full allocated block.

gal_data_t * gal_tile_block_check_tiles (gal_data_t *tilesll)
Make a copy of the memory block and fill it with the index of each tile in tilesll (counting from 0). The non-filled areas will have blank values. The output dataset will have a type of GAL_TYPE_INT32 (see Section 11.3.3 [Library data types (type.h)], page 532).

This function can be used when you want to check the coverage of each tile over the allocated block of memory. It is just a wrapper over the gal_tile_block_write_const_value (with withblank set to zero).

void * gal_tile_block_relative_to_other (gal_data_t *tile, gal_data_t *other)
Return the pointer corresponding to the start of the region covered by tile over the other dataset. See the examples in GAL_TILE_PARSE_OPERATE for some example applications of this function.
void gal_tile_block_blank_flag (gal_data_t *tilell, size_t numthreads)
Check if each tile in the list has blank values and update its flag to mark this check and its result (see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542). The operation will be done on numthreads threads.

GAL_TILE_PARSE_OPERATE (IN, OTHER, PARSE_OTHER, CHECK_BLANK, OP)
Parse IN (which can be a tile or a fully allocated block of memory) and do the OP operation on it. OP can be any combination of C expressions. If OTHER!=NULL, OTHER will be interpreted as a dataset and this macro will allow access to its element(s) and it can optionally be parsed while parsing over IN.

If OTHER is a fully allocated block of memory (not a tile), then the same region that is covered by IN within its own block will be parsed (the same starting pixel with the same number of pixels in each dimension). Hence, in this case, the blocks of OTHER and IN must have the same size. When OTHER is a tile it must have the same size as IN and parsing will start from its starting element/pixel. Also, the respective allocated blocks of OTHER and IN (if different) may have different sizes. Using OTHER (along with PARSE_OTHER), this function-like macro will thus enable you to parse and define your own operation on two fixed size regions in one or two blocks of memory. In the latter case, they may have different numeric data types, see Section 4.5 [Numeric data types], page 178).

The input arguments to this macro are explained below, the expected type of each argument are also written following the argument name:

IN (gal_data_t)
Input dataset, this can be a tile or an allocated block of memory.

OTHER (gal_data_t)
Dataset (gal_data_t) to parse along with IN. It can be NULL. In that case, o (see description of OP below) will be NULL and should not be used. If PARSE_OTHER is zero, only its first element will be used and the size of this dataset is irrelevant.

When OTHER is a block of memory, it has to have the same size as the allocated block of IN. When its a tile, it has to have the same size as IN.

PARSE_OTHER (int)
Parse the other dataset along with the input. When this is non-zero and OTHER!=NULL, then the o pointer will be incremented to cover the OTHER tile at the same rate as i, see description of OP for i and o.

CHECK_BLANK (int)
If it is non-zero, then the input will be checked for blank values and OP will only be called when we are not on a blank element.

OP
Operator: this can be any number of C expressions. This macro is going to define a itype *i variable which will increment over each element of the input array/tile. itype will be replaced with the C type that corresponds to the type of INPUT. As an example, if INPUT's type is
GAL_DATA_UINT16 or GAL_DATA_FLOAT32, i will be defined as uint16 or float respectively.

This function-like macro will also define an otype *o which you can use to access an element of the OTHER dataset (if OTHER!=NULL). o will correspond to the type of OTHER (similar to itype and INPUT discussed above). If PARSE_OTHER is non-zero, then o will also be incremented to the same index element but in the other array. You can use these along with any other variable you define before this macro to process the input and/or the other.

All variables within this function-like macro begin with tpo_ except for the three variables listed below. Therefore, as long as you don’t start the names of your variables with this prefix everything will be fine. Note that i (and possibly o) will be incremented once by this function-like macro, so don’t increment them within OP.

i  Pointer to the element of INPUT that is being parsed with the proper type.

o  Pointer to the element of OTHER that is being parsed with the proper type. o can only be used if OTHER!=NULL and it will be parsed/incremented if PARSE_OTHER is non-zero.

b  Blank value in the type of INPUT.

You can use a given tile (tile on a dataset that it was not initialized with but has the same size, let’s call it new) with the following steps:

```c
void *tarray;
gal_data_t *tblock;

/* 'tile->block' must be corrected AFTER 'tile->array'. */
tarray  = tile->array;
tblock  = tile->block;
tile->array = gal_tile_block_relative_to_other(tile, new);
tile->block = new;

/* Parse and operate over this region of the 'new' dataset. */
GAL_TILE_PARSE_OPERATE(tile, NULL, 0, 0, {
    YOUR_PROCESSING;
});

/* Reset 'tile->block' and 'tile->array'. */
tile->array=tarray;
tile->block=tblock;
```

You can work on the same region of another block in one run of this function-like macro. To do that, you can make a fake tile and pass that as the OTHER argument. Below is a demonstration, tile is the actual tile that you start with and new is the other block of allocated memory.

```c
size_t zero=0;
```
Chapter 11: Library

11.3.15.2 Tile grid

One very useful application of tiles is to completely cover an input dataset with tiles. Such that you know every pixel/data-element of the input image is covered by only one tile. The constructs in this section allow easy definition of such a tile structure. They will create lists of tiles that are also usable by the general tools discussed in Section 11.3.15.1 [Independent tiles], page 609.

As discussed in Section 4.8 [Tessellation], page 189, (mainly raw) astronomical images will mostly require two layers of tessellation, one for amplifier channels which all have the same size and another (smaller tile-size) tessellation over each channel. Hence, in this section we define a general structure to keep the main parameters of this two-layer tessellation and help in benefiting from it.

**gal_tile_two_layer_params**

The general structure to keep all the necessary parameters for a two-layer tessellation.

```c
struct gal_tile_two_layer_params {
    /* Inputs */
    size_t *tilesize;    /* These parameters have to be */
    size_t *numchannels; /* filled manually before */
    float remainderfrac; /* calling the functions in */
    uint8_t workoverch;  /* These parameters have to be */
}
```
uint8_t checktiles; /* this section. */
uint8_t oneelempertile; /***********************************************************/

/* Internal parameters. */
size_t ndim;
size_t tottiles;
size_t tottilesinch;
size_t totchannels;
size_t *channelsize;
size_t *numtiles;
size_t *numtilesinch;
char *tilecheckname;
size_t *permutation;
size_t *firsttsize;

/* Tile and channel arrays (which are also lists). */
gal_data_t *tiles;
gal_data_t *channels;
}

size_t * gal_tile_full (gal_data_t *input, size_t *regular, float remainderfrac,
gal_data_t **out, size_t multiple, size_t **firsttsize)
Cover the full dataset with (mostly) identical tiles and return the number of tiles created along each dimension. The regular tile size (along each dimension) is determined from the regular array. If input’s size is not an exact multiple of regular for each dimension, then the tiles touching the edges in that dimension will have a different size to fully cover every element of the input (depending on remainderfrac).

The output is an array with the same dimensions as input which contains the number of tiles along each dimension. See Section 4.8 [Tessellation], page 189, for a description of its application in Gnuastro’s programs and remainderfrac, just note that this function defines only one layer of tiles.

This is a low-level function (independent of the gal_tile_two_layer_params structure defined above). If you want a two-layer tessellation, directly call gal_tile_full_two_layers that is described below. The input arguments to this function are:

input The main dataset (allocated block) which you want to create a tessellation over (only used for its sizes). So input may be a tile also.

regular The size of the regular tiles along each of the input’s dimensions. So it must have the same number of elements as the dimensions of input (or input->ndim).

remainderfrac The significant fraction of the remainder space to see if it should be split into two and put on both sides of a dimension or not. This is thus only relevant input length along a dimension isn’t an exact multiple of
the regular tile size along that dimension. See Section 4.8 [Tessellation], page 189, for a more thorough discussion.

**out**  Pointer to the array of data structures that will keep all the tiles (see Section 11.3.6.3 [Arrays of datasets], page 548). If *out==NULL, then the necessary space to keep all the tiles will be allocated. If not, then all the tile information will be filled from the dataset that *out points to, see multiple for more.

**multiple**  When *out==NULL (and thus will be allocated by this function), allocate space for multiple times the number of tiles needed. This can be very useful when you have several more identically sized inputs, and you want all their tiles to be allocated (and thus indexed) together, even though they have different block datasets (that then link to one allocated space). See the definition of channels in Section 4.8 [Tessellation], page 189, and gal_tile_full_two_layers below.

**firsttsize**  The size of the first tile along every dimension. This is only different from the regular tile size when regular is not an exact multiple of input’s length along every dimension. This array is allocated internally by this function.

```c
void gal_tile_full_sanity_check (char *filename, char *hdu, gal_data_t *input,
                               struct gal_tile_two_layer_params *tl)
```

Make sure that the input parameters (in tl, short for two-layer) correspond to the input dataset. filename and hdu are only required for error messages. Also, allocate and fill the tl->channelsize array.

```c
void gal_tile_full_two_layers (gal_data_t *input, struct gal_tile_two_layer_params
                              *tl)
```

Create the two layered tessellation in tl. The general set of steps you need to take to define the two-layered tessellation over an image can be seen in the example code below.

```c
gal_data_t *input;
struct gal_tile_two_layer_params tl;
char *filename="input.fits", *hdu="1";

/* Set all the inputs shown in the structure definition. */
...

/* Read the input dataset. */
input=gal_fits_img_read(filename, hdu, -1, 1);

/* Do a sanity check and preparations. */
gal_tile_full_sanity_check(filename, hdu, input, &tl);
```
/* Build the two-layer tessellation*/
gal_tile_full_two_layers(input, &tl);

/* ‘tl.tiles’ and ‘tl.channels’ are now a lists of tiles.*/

void gal_tile_full_permutation (struct gal_tile_two_layer_params *tl)

Make a permutation to allow the conversion of tile location in memory to its location in the full input dataset and put it in tl->permutation. If a permutation has already been defined for the tessellation, this function will not do anything. If permutation won’t be necessary (there is only one channel or one dimension), then this function will not do anything (tl->permutation must have been initialized to NULL).

When there is only one channel OR one dimension, the tiles are allocated in memory in the same order that they represent the input data. However, to make channel-independent processing possible in a generic way, the tiles of each channel are allocated contiguously. So, when there is more than one channel AND more than one dimension, the index of the tile does not correspond to its position in the grid covering the input dataset.

The example below may help clarify: assume you have a 6x6 tessellation with two channels in the horizontal and one in the vertical. On the left you can see how the tile IDs correspond to the input dataset. NOTE how ‘03’ is on the second row, not on the first after ‘02’. On the right, you can see how the tiles are stored in memory (and shown if you simply write the array into a FITS file for example).

<table>
<thead>
<tr>
<th>Corresponding to input</th>
<th>In memory</th>
</tr>
</thead>
<tbody>
<tr>
<td>15 16 17 33 34 35</td>
<td>30 31 32 33 34 35</td>
</tr>
<tr>
<td>12 13 14 30 31 32</td>
<td>24 25 26 27 28 29</td>
</tr>
<tr>
<td>09 10 11 27 28 29</td>
<td>18 19 20 21 22 23</td>
</tr>
<tr>
<td>06 07 08 24 25 26</td>
<td>12 13 14 15 16 17</td>
</tr>
<tr>
<td>03 04 05 21 22 23</td>
<td>06 07 08 09 10 11</td>
</tr>
<tr>
<td>00 01 02 18 19 20</td>
<td>00 01 02 03 04 05</td>
</tr>
</tbody>
</table>

As a result, if your values are stored in same order as the tiles, and you want them in over-all memory (for example to save as a FITS file), you need to permute the values:

```
    gal_permutation_apply(values, tl->permutation);
```

If you have values over-all and you want them in tile-order, you can apply the inverse permutation:

```
    gal_permutation_apply_inverse(values, tl->permutation);
```

Recall that this is the definition of permutation in this context:

```
    permute: IN_ALL[ i ] = IN_MEMORY[ perm[i] ]
    inverse: IN_ALL[ perm[i] ] = IN_MEMORY[ i ]
```
void

gal_tile_full_values_write (gal_data_t *tilevalues, struct
gal_tile_two_layer_params *tl, int withblank, char *filename,
gal_fits_list_key_t *keys, char *program_string)

Write one value for each tile into a file. It is important to note that the values in tilevalues must be ordered in the same manner as the tiles, so tilevalues->array[i] is the value that should be given to tl->tiles[i]. The tl->permutation array must have been initialized before calling this function with gal_tile_full_permutation.

If withblank is non-zero, then block structure of the tiles will be checked and all blank pixels in the block will be blank in the final output file also.

gal_data_t *

gal_tile_full_values_smooth (gal_data_t *tilevalues, struct
gal_tile_two_layer_params *tl, size_t width, size_t numthreads)

Smooth the given values with a flat kernel of the given width. This cannot be done manually because if tl->workoverch==0, tiles in different channels must not be mixed/smoothed. Also the tiles are contiguous within the channel, not within the image, see the description under gal_tile_full_permutation.

size_t

gal_tile_full_id_from_coord (struct gal_tile_two_layer_params *tl, size_t *coord)

Return the ID of the tile that corresponds to the coordinates coord. Having this ID, you can use the tl->tiles array to get to the proper tile or read/write a value into an array that has one value per tile.

void

gal_tile_full_free_contents (struct gal_tile_two_layer_params *tl)

Free all the allocated arrays within tl.

11.3.16 Bounding box (box.h)

Functions related to reporting the bounding box of certain inputs are declared in gnuastro/box.h. All coordinates in this header are in the FITS format (first axis is the horizontal and the second axis is vertical).

void

gal_box_bound_ellipse_extent (double a, double b, double theta_deg, double *
extent)

Return the maximum extent along each dimension of the given ellipse from the center of the ellipse. Therefore this is half the extent of the box in each dimension. a is the ellipse semi-major axis, b is the semi-minor axis, theta_deg is the position angle in degrees. The extent in each dimension is in floating point format and stored in extent which must already be allocated before this function.

void

gal_box_bound_ellipse (double a, double b, double theta_deg, long *width)

Any ellipse can be enclosed into a rectangular box. This function will write the height and width of that box where width points to. It assumes the center of the ellipse is
located within the central pixel of the box. \( a \) is the ellipse semi-major axis length, \( b \) is the semi-minor axis, \( \theta_{\deg} \) is the position angle in degrees. The \( \text{width} \) array will contain the output size in long integer type. \( \text{width}[0] \) and \( \text{width}[1] \) are the number of pixels along the first and second FITS axis. Since the ellipse center is assumed to be in the center of the box, all the values in \( \text{width} \) will be an odd integer.

```c
void gal_box_bound_ellipsoid_extent (double *semiaxes, double *euler_deg,
                                    double *extent)
Return the maximum extent along each dimension of the given ellipsoid from its
center. Therefore this is half the extent of the box in each dimension. The semi-axis
lengths of the ellipsoid must be present in the 3 element \( \text{semiaxis} \) array. The \( \text{euler_deg} \) array contains the three ellipsoid Euler angles in degrees. For a description of
the Euler angles, see description of \( \text{gal_box_bound_ellipsoid} \) below. The extent in
each dimension is in floating point format and stored in \( \text{extent} \) which must already
be allocated before this function.
```

```c
void gal_box_bound_ellipsoid (double *semiaxes, double *euler_deg, long *width)
Any ellipsoid can be enclosed into a rectangular volume/box. The purpose of this
function is to give the integer size/width of that box. The semi-axes lengths of the
ellipse must be in the \( \text{semiaxes} \) array (with three elements). The major axis length
must be the first element of \( \text{semiaxes} \). The only other condition is that the next two
semi-axes must both be smaller than the first. The orientation of the major axis is
defined through three proper Euler angles (ZXZ order in degrees) that are given in
the \( \text{euler_deg} \) array. The \( \text{width} \) array will contain the output size in long integer
type (in FITS axis order). Since the ellipsoid center is assumed to be in the center of
the box, all the values in \( \text{width} \) will be an odd integer.

The proper Euler angles can be defined in many ways (which axes to rotate about).
For a full description of the Euler angles, please see Wikipedia (https://en.
wikipedia.org/wiki/Euler_angles). Here we adopt the ZXZ (or \( Z_1X_2Z_3 \)) proper
Euler angles were the first rotation is done around the \( Z \) axis, the second one about
the (rotated) \( X \) axis and the third about the (rotated) \( Z \) axis.
```

```c
void gal_box_border_from_center (double center, size_t ndim, long *width, long
                                *fpixel, long *lpixel)
Given the center coordinates in \( \text{center} \) and the \( \text{width} \) (along each dimension) of a
box, return the coordinates of the first (\( \text{fpixel} \) and last (\( \text{lpixel} \) pixels. All arrays
must have \( \text{ndim} \) elements (one for each dimension).
```

```c
int gal_box_overlap (long *naxes, long *fpixel_i, long *lpixel_i, long
                     *fpixel_o, long *lpixel_o, size_t ndim)
An \( \text{ndim} \)-dimensional dataset of size \( \text{naxes} \) (along each dimension, in FITS order)
and a box with first and last (inclusive) coordinate of \( \text{fpixel_i} \) and \( \text{lpixel_i} \) is
given. This box doesn’t necessarily have to lie within the dataset, it can be outside
of it, or only partially overlap. This function will change the values of \texttt{fpixel\_i} and \texttt{lpixel\_i} to exactly cover the overlap in the input dataset’s coordinates.

This function will return 1 if there is an overlap and 0 if there isn’t. When there is an overlap, the coordinates of the first and last pixels of the overlap will be put in \texttt{fpixel\_o} and \texttt{lpixel\_o}.

### 11.3.17 Polygons (\texttt{polygon.h})

Polygons are commonly necessary in image processing. For example in Crop they are used for cutting out non-rectangular regions of an image (see Section 6.1 [Crop], page 259), and in Warp, for mapping different pixel grids over each other (see Section 6.4 [Warp], page 328).

Polygons come in two classes: convex and concave (or generally, non-convex!), see below for a demonstration. Convex polygons are those where all inner angles are less than 180 degrees. By contrast, a convex polygon is one where an inner angle may be more than 180 degrees.

<table>
<thead>
<tr>
<th>Concave Polygon</th>
<th>Convex Polygon</th>
</tr>
</thead>
<tbody>
<tr>
<td>D -------- C</td>
<td>D------------- C</td>
</tr>
<tr>
<td>\ /</td>
<td>\ /</td>
</tr>
<tr>
<td>\ \</td>
<td>\ \</td>
</tr>
<tr>
<td>A---------- B</td>
<td>A ---------- B</td>
</tr>
</tbody>
</table>

In all the functions here the vertices (and points) are defined as an array. So a polygon with 4 vertices will be identified with an array of 8 elements with the first two elements keeping the 2D coordinates of the first vertice and so on.

**GAL\_POLYGON\_MAX\_CORNERS**

The largest number of vertices a polygon can have in this library.

**GAL\_POLYGON\_ROUND\_ERR**

We have to consider floating point round-off errors when dealing with polygons. For example we will take \texttt{A} as the maximum of \texttt{A} and \texttt{B} when \texttt{A>B-GAL\_POLYGON\_ROUND\_ERR}.

**void gal\_polygon\_vertices\_sort\_convex (double *in, size\_t n, size\_t *ordinds)**

We have a simple polygon (that can result from projection, so its edges don’t collide or it doesn’t have holes) and we want to order its corners in an anticlockwise fashion. This is necessary for clipping it and finding its area later. The input vertices can have practically any order.

The input (\texttt{in}) is an array containing the coordinates (two values) of each vertice. \texttt{n} is the number of corners. So \texttt{in} should have \texttt{2*n} elements. The output (\texttt{ordinds}) is an array with \texttt{n} elements specifying the indices in order. This array must have been allocated before calling this function. The indexes are output for more generic usage, for example in a homographic transform (necessary in warping an image, see Section 6.4.1 [Warping basics], page 329), the necessary order of vertices is the same for all the pixels. In other words, only the positions of the vertices change, not the way they need to be ordered. Therefore, this function would only be necessary once.
As a summary, the input is unchanged, only n values will be put in the \texttt{ordinds} array. Such that calling the input coordinates in the following fashion will give an anti-clockwise order when there are 4 vertices:

1st vertex: \texttt{in[ordinds[0]*2, in[ordinds[0]*2+1]}
2nd vertex: \texttt{in[ordinds[1]*2, in[ordinds[1]*2+1]}
3rd vertex: \texttt{in[ordinds[2]*2, in[ordinds[2]*2+1]}
4th vertex: \texttt{in[ordinds[3]*2, in[ordinds[3]*2+1]}

The implementation of this is very similar to the Graham scan in finding the Convex Hull. However, in projection we will never have a concave polygon (the left condition below, where this algorithm will get to E before D), we will always have a convex polygon (right case) or E won’t exist! This is because we are always going to be calculating the area of the overlap between a quadrilateral and the pixel grid or the quadrilateral itself.

The \texttt{GAL\_POLYGON\_MAX\_CORNERS} macro is defined so there will be no need to allocate these temporary arrays separately. Since we are dealing with pixels, the polygon can’t really have too many vertices.

\begin{verbatim}
int gal_polygon_is_convex (double *v, size_t n)
    Returns 1 if the polygon is convex with vertices defined by v and 0 if it is a concave polygon. Note that the vertices of the polygon should be sorted in an anti-clockwise manner.

double gal_polygon_area (double *v, size_t n)
    Find the area of a polygon with vertices defined in v. v points to an array of doubles which keep the positions of the vertices such that v[0] and v[1] are the positions of the first vertex to be considered.

int gal_polygon_is_inside (double *v, double *p, size_t n)
    Returns 0 if point p in inside a polygon, either convex or concave. The vertices of the polygon are defined by v and 0 otherwise, they have to be ordered in an anti-clockwise manner. This function uses the winding number algorithm (https://en.wikipedia.org/wiki/Point_in_polygon#Winding_number_algorithm) to check the points. Note that this is a generic function (working on both concave and convex polygons, so if you know before-hand that your polygon is convex, it is much more efficient to use \texttt{gal\_polygon\_is\_inside\_convex}.

int gal_polygon_is_inside_convex (double *v, double *p, size_t n)
    Return 1 if the point p is within the polygon whose vertices are defined by v. The polygon is assumed to be convex, for a more generic function that deals with concave and convex polygons, see \texttt{gal\_polygon\_is\_inside}. Note that the vertices of the polygon have to be sorted in an anti-clock-wise manner.
\end{verbatim}
int [Function]
gal_polygon_ppropin (double *v, double *p, size_t n)
Similar to gal_polygon_is_inside_convex, except that if the point p is on one of
the edges of a polygon, this will return 0.

int [Function]
gal_polygon_is_counterclockwise (double *v, size_t n)
Returns 1 if the sorted polygon has a counter-clockwise orientation and 0 otherwise.
This function uses the concept of “winding”, which defines the relative order in which
the vertices of a polygon are listed to determine the orientation of vertices. For
complex polygons (where edges, or sides, intersect), the most significant orientation
is returned. In a complex polygon, when the alternative windings are equal (for
example an 8-shape) it will return 1 (as if it was counter-clockwise). Note that the
polygon vertices have to be sorted before calling this function.

int [Function]
gal_polygon_to_counterclockwise (double *v, size_t n)
Arrange the vertices of the sorted polygon in place, to be in a counter-clockwise
direction. If the input polygon already has a counter-clockwise direction it won’t
touch the input. The return value is 1 on successful execution. This function is just a
wrapper over gal_polygon_is_counterclockwise, and will reverse the order of the
vertices when necessary.

void [Function]
gal_polygon_clip (double *s, size_t n, double *c, size_t m, double *o, size_t
*numcrn)
Clip (find the overlap of) two polygons. This function uses the Sutherland-Hodgman
https://en.wikipedia.org/wiki/Sutherland%E2%80%93Hodgman_algorithm
polygon clipping algorithm. Note that the vertices of both polygons have to be
sorted in an anti-clock-wise manner.

The Pseudocode from Wikipedia:
List outputList = subjectPolygon;
for (Edge clipEdge in clipPolygon) do
    List inputList = outputList;
    outputList.clear();
    Point S = inputList.last;
    for (Point E in inputList) do
        if (E inside clipEdge) then
            if (S not inside clipEdge) then
                outputList.add(ComputeIntersection(S,E,clipEdge));
            end if
            outputList.add(E);
        else if (S inside clipEdge) then
            outputList.add(ComputeIntersection(S,E,clipEdge));
        end if
        S = E;
    done

The difference is that we are not using lists, but arrays to keep polygon vertices. The two polygons are called Subject $s$ and Clip $c$ with $n$ and $m$ vertices respectively. The output is stored in $o$ and the number of elements in the output are stored in what *numcrn* (for number of corners) points to.

**function**

```c
void gal_polygon_vertices_sort (double *vertices, size_t n, size_t *ordinds)
```

Sort the indices of the un-ordered vertices array to a counter-clockwise polygon in the already allocated space of ordinds. It is assumed that there are $n$ vertices, and thus that vertices contains $2*n$ elements where the two coordinates of the first vertice occupy the first two elements of the array and so on.

The polygon can be both concave and convex (see the start of this section). However, note that for concave polygons there is no unique sort from an un-ordered set of vertices. So after this function you may want to use **gal_polygon_is_convex** and print a warning to check the output if the polygon was concave.

Note that the contents of the vertices array are left untouched by this function. If you want to write the ordered vertice coordinates in another array with the same size, you can use a loop like this:

```c
for(i=0;i<n;++i)
{
    ordered[i*2  ] = vertices[  ordinds[i]*2   ];
    ordered[i*2+1] = vertices[  ordinds[i]*2 + 1];
}
```

In this algorithm, we find the rightmost and leftmost points (based on their x-coordinate) and use the diagonal vector between those points to group the points in arrays based on their position with respect to this vector. For anticlockwise sorting, all the points below the vector are sorted by their ascending x-coordinates and points above the vector are sorted in decreasing order using **qsort**. Finally, both these arrays are merged together to get the final sorted array of points, from which the points are indexed into the ordinds using linear search.

### 11.3.18 Qsort functions (**qsort.h**)

When sorting a dataset is necessary, the C programming language provides the **qsort** (Quick sort) function. **qsort** is a generic function which allows you to sort any kind of data structure (not just a single array of numbers). To define “greater” and “smaller” (for sorting), **qsort** needs another function, even for simple numerical types. The functions introduced in this section are to passed onto **qsort**.

Note that larger and smaller operators are not defined on NaN elements. Therefore, if the input array is a floating point type, and contains NaN values, the relevant functions of this section are going to put the NaN elements at the end of the list (after the sorted non-NaN elements), irrespective of the requested sorting order (increasing or decreasing).

The first class of functions below (with **TYPE** in their names) can be used for sorting a simple numeric array. Just replace **TYPE** with the dataset’s numeric datatype. The second set of functions can be used to sort indices (leave the actual numbers untouched). To use
the second set of functions, a global variable or structure are also necessary as described below.

\textbf{gal\_qsort\_index\_single} \hspace{1cm} \textbf{[Global variable]}

Pointer to an array (for example float * or int *) to use as a reference in \texttt{gal\_qsort\_index\_single\_TYPE\_d} or \texttt{gal\_qsort\_index\_single\_TYPE\_i}, see the explanation of these functions for more. Note that if more than one array is to be sorted in a multi-threaded operation, these functions will not work as expected. However, when all the threads just sort the indices based on a single array, this global variable can safely be used in a multi-threaded scenario.

\textbf{gal\_qsort\_index\_multi} \hspace{1cm} \textbf{[Type (C struct)]}

Structure to get the sorted indices of multiple datasets on multiple threads with \texttt{gal\_qsort\_index\_multi\_d} or \texttt{gal\_qsort\_index\_multi\_i}. Note that the values array will not be changed by these functions, it is only read. Therefore all the values elements in the (to be sorted) array of \texttt{gal\_qsort\_index\_multi} must point to the same place.

\begin{verbatim}
struct gal\_qsort\_index\_multi
{
    float *values; /* Array of values (same in all). */
    size_t index; /* Index of each element to be sorted. */
};
\end{verbatim}

\textbf{int \texttt{gal\_qsort\_TYPE\_d} (const void *a, const void *b)}

When passed to \texttt{qsort}, this function will sort a \texttt{TYPE} array in decreasing order (first element will be the largest). Please replace \texttt{TYPE} (in the function name) with one of the Section 4.5 [Numeric data types], page 178, for example \texttt{gal\_qsort\_int32\_d}, or \texttt{gal\_qsort\_float64\_d}.

\textbf{int \texttt{gal\_qsort\_TYPE\_i} (const void *a, const void *b)}

When passed to \texttt{qsort}, this function will sort a \texttt{TYPE} array in increasing order (first element will be the smallest). Please replace \texttt{TYPE} (in the function name) with one of the Section 4.5 [Numeric data types], page 178, for example \texttt{gal\_qsort\_int32\_i}, or \texttt{gal\_qsort\_float64\_i}.

\textbf{int \texttt{gal\_qsort\_index\_single\_TYPE\_d} (const void *a, const void *b)}

When passed to \texttt{qsort}, this function will sort a \texttt{size_t} array based on decreasing values in the \texttt{gal\_qsort\_index\_single}. The global \texttt{gal\_qsort\_index\_single} pointer has a void * pointer which will be cast to the proper type based on this function: for example \texttt{gal\_qsort\_index\_single\_uint16\_d} will cast the array to an unsigned 16-bit integer type. The array that \texttt{gal\_qsort\_index\_single} points to will not be changed, it is only read. For example, see this demo program:

\begin{verbatim}
#include <stdio.h>
#include <stdlib.h> /* qsort is defined in stdlib.h. */
#include <gnuastro/qsort.h>
\end{verbatim}
int main (void)
{
    size_t s[4]={0, 1, 2, 3};
    float f[4]={1.3,0.2,1.8,0.1};
    gal_qsort_index_single=f;
    qsort(s, 4, sizeof(size_t), gal_qsort_index_single_float_d);
    printf("%zu, %zu, %zu, %zu\n", s[0], s[1], s[2], s[3]);
    return EXIT_SUCCESS;
}

The output will be: 2, 0, 1, 3.

int gal_qsort_index_single_TYPE_i (const void *a, const void *b)
Similar to gal_qsort_index_single_TYPE_d, but will sort the indexes such that the values of gal_qsort_index_single can be parsed in increasing order.

int gal_qsort_index_multi_d (const void *a, const void *b)
When passed to qsort with an array of gal_qsort_index_multi, this function will sort the array based on the values of the given indices. The sorting will be ordered according to the values pointer of gal_qsort_index_multi. Note that values must point to the same place in all the structures of the gal_qsort_index_multi array.
This function is only useful when the indices of multiple arrays on multiple threads are to be sorted. If your program is single threaded, or all the indices belong to a single array (sorting different sub-sets of indices in a single array on multiple threads), it is recommended to use gal_qsort_index_single_d.

int gal_qsort_index_multi_i (const void *a, const void *b)
Similar to gal_qsort_index_multi_d, but the result will be sorted in increasing order (first element will have the smallest value).

11.3.19 K-d tree (kdtree.h)
K-d tree is a space-partitioning binary search tree for organizing points in a k-dimensional space. They are a very useful data structure for multidimensional searches like range searches and nearest neighbor searches. For a more formal and complete introduction see the Wikipedia page (https://en.wikipedia.org/wiki/K-d_tree).

Each non-leaf node in a k-d tree divides the space into two parts, known as half-spaces. To select the top/root node for partitioning, we find the median of the points and make a hyperplane normal to the first dimension. The points to the left of this space are represented by the left subtree of that node and points to the right of the space are represented by the right subtree. This is then repeated for all the points in the input, thus associating a “left” and “right” branch for each input point.

Gnuastro uses the standard algorithms of the k-d tree with one small difference that makes it much more memory and CPU optimized. The set of input points that define the
tree nodes are given as a list of Gnuastro’s data container type, see Section 11.3.8.9 [List of gal_data_t], page 566. Each gal_data_t in the list represents the point’s coordinate in one dimension, and the first element in the list is the first dimension. Hence the number of data values in each gal_data_t (which must be equal in all of them) represents the number of points. This is the same format that Gnuastro’s Table reading/writing functions read/write columns in tables, see Section 11.3.10 [Table input output (table.h)], page 568.

The output k-d tree is a list of two gal_data_ts, representing the input’s row-number (or index, counting from 0) of the left and right subtrees of each row. Each gal_data_t thus has the same number of rows (or points) as the input, but only containing integers with a type of uint32_t (unsigned 32-bit integer). If a node has no left, or right subtree, then GAL_BLANK_UINT32 will be used. Below you can see the simple tree for 2D points from Wikipedia. The input point coordinates are represented as two input gal_data_ts (X and Y, where X->next=Y and Y->next=NULL). If you had three dimensional points, you could define an extra gal_data_t such that Y->next=Z and Z->next=NULL. The output is always a list of two gal_data_ts, where the first one contains the index of the left sub-tree in the input, and the second one, the index of the right subtree. The index of the root node (0 in the case below) is also returned as a single number.

<table>
<thead>
<tr>
<th>INDEX (as guide)</th>
<th>INPUT</th>
<th>OUTPUT</th>
<th>K-D Tree (visualized)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>5</td>
<td>1</td>
<td>(5,4)</td>
</tr>
<tr>
<td>1</td>
<td>2</td>
<td>BLANK</td>
<td>(2,3)</td>
</tr>
<tr>
<td>2</td>
<td>7</td>
<td>5</td>
<td>\</td>
</tr>
<tr>
<td>3</td>
<td>9</td>
<td>BLANK</td>
<td>(7,2)</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>BLANK</td>
<td>\</td>
</tr>
<tr>
<td>5</td>
<td>8</td>
<td>BLANK</td>
<td>(8,1) (9,6)</td>
</tr>
</tbody>
</table>

This format is therefore scalable to any number of dimensions: the number of dimensions are determined from the number of nodes in the input list of gal_data_ts (for example, using gal_list_data_number). In Gnuastro’s k-d tree implementation, there are thus no special structures to keep every tree node (which would take extra memory and would need to be moved around as the tree is being created). Everything is done internally on the index of each point in the input dataset: the only thing that is flipped/sorted during tree creation is the index to the input row for any number of dimensions. As a result, Gnuastro’s k-d tree implementation is very memory and CPU efficient and its two output columns can directly be written into a standard table (without having to define any special binary format).

```
gal_data_t * gal_kdmtree_create (gal_data_t *coords_raw, size_t *root)
  [Function]
  Create a k-d tree in a bottom-up manner (from leaves to the root). This function returns two gal_data_ts connected as a list, see description above. The first dataset contains the indexes of left and right nodes of the subtrees for each input node. The index of the root node is written into the memory that root points to. coords_raw is the list of the input points (one gal_data_t per dimension, see above).
```

---

24 This example input table is the same as the example in Wikipedia (as of December 2020). However, on the Wikipedia output, the root node is (7,2), not (5,4). The difference is primarily because there are 6 rows and the median element of an even number of elements can vary by integer calculation strategies. Here we use 0-based indexes for finding median and round to the smaller integer.
For example, assume you have the simple set of points below (from the visualized example at the start of this section) in a plain-text file called coordinates.txt:

```
$ cat coordinates.txt
5 4
2 3
7 2
9 6
4 7
8 1
```

With the program below, you can calculate the kd-tree, and write it in a FITS file (while keeping the root index as a FITS keyword inside of it).

```c
#include <stdio.h>
#include <gnuastro/table.h>
#include <gnuastro/kdtree.h>

int main (void)
{
    gal_data_t *input, *kdtree;
    char kdtreefile[]="kd-tree.fits";
    char inputfile[]="coordinates.txt";

    /* To write the root within the saved file. */
    size_t root;
    char *unit="index";
    char *keyname="KDTROOT";
    gal_fits_list_key_t *keylist=NULL;
    char *comment="k-d tree root index (counting from 0).";

    /* Read the input table. Note: this assumes the table only
    * contains your input point coordinates (one column for each
    * dimension). If it contains more columns with other properties
    * for each point, you can specify which columns to read by
    * name or number, see the documentation of 'gal_table_read'. */
    input=gal_table_read(inputfile, "1", NULL, NULL,
                        GAL_TABLE_SEARCH_NAME, 0, -1, 0, NULL);

    /* Construct a k-d tree. The index of root is stored in 'root'. */
    kdtree=gal_kdtree_create(input, &root);

    /* Write the k-d tree to a file and write root index and input
    * name as FITS keywords ('gal_table_write' frees 'keylist'). */
    gal_fits_key_list_title_add(&keylist, "k-d tree parameters", 0);
    gal_fits_key_write_filename("KDTIN", inputfile, &keylist, 0);
    gal_fits_key_list_add_end(&keylist, GAL_TYPE_SIZE_T, keyname, 0,
                              &root, 0, comment, 0, unit, 0);
```
gal_table_write(kdtree, &keylist, NULL, GAL_TABLE_FORMAT_BFITS, 
kdtreefile, "kdtree", 0);

/* Clean up and return. */
gal_list_data_free(input);
gal_list_data_free(kdtree);
return EXIT_SUCCESS;
}

You can inspect the saved k-d tree FITS table with Gnuastro’s Section 5.3 [Table], page 230, (first command below), and you can see the keywords containing the root index with Section 5.1 [Fits], page 194, (second command below):

```
asttable kd-tree.fits
astfits kd-tree.fits -h1
```

size_t gal_kdtree_nearest_neighbour (gal_data_t *coords_raw, gal_data_t *kdtree, 
size_t root, double *point, double *least_dist)

Returns the index of the nearest input point to the query point (point, assumed to be an array with same number of elements as gal_data_ts in coords_raw). The distance between the query point and its nearest neighbor is stored in the space that least_dist points to. This search is efficient due to the constant checking for the presence of possible best points in other branches. If it isn’t possible for the other branch to have a better nearest neighbor, that branch is not searched.

As an example, let’s use the k-d tree that was created in the example of gal_kdtree_create (above) and find the nearest row to a given coordinate (point). This will be a very common scenario, especially in large and multi-dimensional datasets where the k-d tree creation can take long and you don’t want to re-create the k-d tree every time. In the gal_kdtree_create example output, we also wrote the k-d tree root index as a FITS keyword (KDTROOT), so after loading the two table data (input coordinates and k-d tree), we’ll read the root from the FITS keyword. This is a very simple example, but the scalability is clear: for example it is trivial to parallelize (see Section 11.4.3 [Library demo - multi-threaded operation], page 664).

```
#include <stdio.h>
#include <gnuastro/table.h>
#include <gnuastro/kdtree.h>

int
main (void)
{
    /* INPUT: desired point. */
    double point[2]={8.9,5.9};

    /* Same as example in description of 'gal_kdtree_create'. */
    gal_data_t *input, *kdtree;
    char kdtreefile[]="kd-tree.fits";
    char inputfile[]="coordinates.txt";
```
/ * Processing variables of this function. */
char kdtreehdu[]="1";
double *in_x, *in_y, least_dist;
size_t root, nkeys=1, nearest_index;
gal_data_t *rkey, *keysll=gal_data_array_calloc(nkeys);

/* Read the input coordinates, see comments in example of
 'gal_kdtree_create' for more. */
input=gal_table_read(inputfile, "1", NULL, NULL,
             GAL_TABLE_SEARCH_NAME, 0, -1, 0, NULL);

/* Read the k-d tree contents (created before). */
kdtree=gal_table_read(kdtreefile, "1", NULL, NULL,
             GAL_TABLE_SEARCH_NAME, 0, -1, 0, NULL);

/* Read the k-d tree root index from the header keyword.
 See example in description of 'gal_fits_key_read_from_ptr'. */
keysll[0].name="KDTROOT";
keysll[0].type=GAL_TYPE_SIZE_T;
gal_fits_key_read(kdtreefile, kdtreehdu, keysll, 0, 0);
keysll[0].name=NULL; /* Since we didn't allocate it. */
rkey=gal_data_copy_to_new_type(&keysll[0], GAL_TYPE_SIZE_T);
root=((size_t *)(rkey->array))[0];

/* Find the nearest neighbour of the point. */
nearest_index=gal_kdtree_nearest_neighbour(input, kdtree, root,
             point, &least_dist);

/* Print the results. */
in_x=input->array;
in_y=input->next->array;
printf("(%g, %g): nearest is (%g, %g), with a distance of %g
",
   point[0], point[1], in_x[nearest_index],
in_y[nearest_index], least_dist);

/* Clean up and return. */
gal_data_free(rkey);
gal_list_data_free(input);
gal_list_data_free(kdtree);
gal_data_array_free(keysll, nkeys, 1);
return EXIT_SUCCESS;
11.3.20 Permutations (permutation.h)

Permutation is the technical name for re-ordering of values. The need for permutations occurs a lot during (mainly low-level) processing. To do permutation, you must provide two inputs: an array of values (that you want to re-order in place) and a permutation array which contains the new index of each element (let’s call it perm). The diagram below shows the input array before and after the re-ordering.

\[
\text{permute: } \text{AFTER}[i] = \text{BEFORE}[\text{perm}[i]] \quad i = 0 \ldots N-1
\]

\[
\text{inverse: } \text{AFTER}[\text{perm}[i]] = \text{BEFORE}[i] \quad i = 0 \ldots N-1
\]

The functions here are a re-implementation of the GNU Scientific Library’s gsl_permute function. The reason we didn’t use that function was that it uses system-specific types (like long and int) which can have different widths on different systems, hence are not easily convertible to Gnuastro’s fixed width types (see Section 4.5 [Numeric data types], page 178). There is also a separate function for each type, heavily using macros to allow a base function to work on all the types. Thus it is hard to read/understand. Hence, Gnuastro contains a re-write of their steps in a new type-agnostic method which is a single function that can work on any type.

As described in GSL’s source code and manual, this implementation comes from Donald Knuth’s *Art of computer programming* book, in the “Sorting and Searching” chapter of Volume 3 (3rd ed). Exercise 10 of Section 5.2 defines the problem and in the answers, Knuth describes the solution. So if you are interested, please have a look there for more.

We are in contact with the GSL developers and in the future we will submit these implementations to GSL. If they are finally incorporated there, we will delete this section in future versions.

\[\text{void gal_permutation_check} (\text{size_t} *\text{permutation}, \text{size_t} \text{size})\]

Print how \text{permutation} will re-order an array that has \text{size} elements for each element in one one line.

\[\text{void gal_permutation_apply} (\text{gal_data_t} *\text{input}, \text{size_t} *\text{permutation})\]

Apply \text{permutation} on the \text{input} dataset (can have any type), see above for the definition of permutation.

\[\text{void gal_permutation_apply_inverse} (\text{gal_data_t} *\text{input}, \text{size_t} *\text{permutation})\]

Apply the inverse of \text{permutation} on the \text{input} dataset (can have any type), see above for the definition of permutation.

11.3.21 Matching (match.h)

Matching is often necessary when two measurements of the same points have been done using different instruments (or hardware), different software or different configurations of the same software. In other words, you have two catalogs or tables, and each has N columns

\[\text{Gnuastro’s Task 14497} (\text{http://savannah.gnu.org/task/14497}).\] If this task is still “postponed” when you are reading this and you are interested to help, your contributions would be very welcome. Both Gnuastro and GSL developers are very busy, hence both would appreciate your help.
containing the N-dimensional “coordinate” values of each point. Each table can have other columns too, for example one can have brightness measurements in one filter, and another can have morphology measurements or etc.

The matching functions here will use the coordinate columns of the two tables to find a permutation for each, and the total number of matched rows ($N_{\text{match}}$). This will enable you to match by the positions if you like. At a higher level, you can apply the permutation to the brightness or morphology columns to merge the catalogs over the $N_{\text{match}}$ rows. The input and output data formats of the functions are the same and described below before the actual functions. Each function also has extra arguments due to the particular algorithm it uses for the matching.

The two inputs of the functions ($\text{coord1}$ and $\text{coord2}$) must be Section 11.3.8.9 [List of gal_data_t], page 566. Each gal_data_t node in $\text{coord1}$ or $\text{coord2}$ should be a single dimensional dataset (column in a table) and all the nodes (in each) must have the same number of elements (rows). In other words, each column can be visualized as having the coordinates of each point in its respective dimension. The dimensions of the coordinates is determined by the number of gal_data_t nodes in the two input lists (which must be equal). The number of rows (or the number of elements in each gal_data_t) in the columns of $\text{coord1}$ and $\text{coord2}$ can (and, usually will!) be different. In summary, these functions will be happy if you use gal_table_read to read the two coordinate columns from a file, see Section 11.3.10 [Table input output (table.h)], page 568.

The functions below return a simply-linked list of three 1D datasets (see Section 11.3.8.9 [List of gal_data_t], page 566), let’s call the returned dataset ret. The first two (ret and ret->next) are permutations. In other words, the array elements of both have a type of size_t, see Section 11.3.20 [Permutations (permutation.h)], page 630. The third node (ret->next->next) is the calculated distance for that match and its array has a type of double. The number of matches will be put in the space pointed by the nummatched argument. If there wasn’t any match, this function will return NULL.

The two permutations can be applied to the rows of the two inputs: the first one (ret) should be applied to the rows of the table containing coord1 and the second one (ret->next) to the table containing coord2. After applying the returned permutations to the inputs, the top nummatched elements of both will match with each other. The ordering of the rest of the elements is undefined (depends on the matching function used). The third node is the distances between the respective match (which may be elliptical distance, see discussion of “aperture” below).

The functions will not simply return the nearest neighbor as a match. This is because the nearest neighbor may be too far to be a meaningful! They will check the distance between the nearest neighbor of each point and only return a match if it is within an acceptable N-dimensional distance (or “aperture”). The matching aperture is defined by the aperture array that is an input argument to the functions.

If several points of one catalog lie within this aperture of a point in the other catalog, the nearest is defined as the match. In a 2D situation (where the input lists have two nodes), for the most generic case, aperture must have three elements: the major axis length, axis ratio and position angle (see Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440). If aperture[1]==1, the aperture will be a circle of radius aperture[0] and the third value won’t be used. When the aperture is an ellipse, distances between the points are also
calculated in the respective elliptical distances \( r_{el} \) in Section 8.1.1.1 [Defining an ellipse and ellipsoid], page 440).

**Output permutations ignore internal sorting:** the output permutations will correspond to the initial inputs. Therefore, even when \( \text{inplace}! = 0 \) (and this function re-arranges the inputs in place), the output permutation will correspond to original (possibly non-sorted) inputs. The reason for this is that you rarely want to permute the actual positional columns after the match. Usually, you also have other columns (for example the brightness, morphology and etc) and you want to find how they differ between the objects that match. Once you have the permutations, they can be applied to those other columns (see Section 11.3.20 [Permutations (permutation.h)], page 630) and the higher-level processing can continue. So if you don’t need the coordinate columns for the rest of your analysis, it is better to set \( \text{inplace}=1 \).

**Function**

```c
gal_data_t * gal_match_sort_based (gal_data_t *coord1, gal_data_t *coord2, double *aperture, int sorted_by_first, int inplace, size_t minmapsize, int quietmmap, size_t *nummatched)
```

Use a basic sort-based match to find the matching points of two input coordinates. See the descriptions above on the format of the inputs and outputs. To speed up the search, this function will sort the input coordinates by their first column (first axis). If both are already sorted by their first column, you can avoid the sorting step by giving a non-zero value to \( \text{sorted_by_first} \).

When sorting is necessary and \( \text{inplace} \) is non-zero, the actual input columns will be sorted. Otherwise, an internal copy of the inputs will be made, used (sorted) and later freed before returning. Therefore, when \( \text{inplace}=0 \), inputs will remain untouched, but this function will take more time and memory. If internal allocation is necessary and the space is larger than \( \text{minmapsize} \), the space will be not allocated in the RAM, but in a file, see description of \(--\text{minmapsize}\) and \(--\text{quietmmap}\) in Section 4.1.2.2 [Processing options], page 159.

**Function**

```c
gal_data_t * gal_match_kdtree (gal_data_t *coord1, gal_data_t *coord2, gal_data_t *coord1_kdtree, size_t kdtree_root, double *aperture, size_t numthreads, size_t minmapsize, int quietmmap, size_t *nummatched)
```

Use the k-d tree concept for finding matches between two catalogs, optionally in parallel (on \( \text{numthreads} \) threads). The k-d tree of the first input (\( \text{coord1}_\text{kdtree} \)), and its root index (\( \text{kdtree}_\text{root} \)), should be constructed and found before calling this function, to do this, you can use the \text{gal_kdtree_create} of Section 11.3.19 [K-d tree (kdtree.h)], page 625. The desired \( \text{aperture} \) array is the same as \text{gal_match_sort_based} and described at the top of this section.

The final number of matches is returned in \( \text{nummatched} \) and the format of the returned dataset (three columns) is described above. If internal allocation is necessary and the space is larger than \( \text{minmapsize} \), the space will be not allocated in the RAM, but in a file, see description of \(--\text{minmapsize}\) and \(--\text{quietmmap}\) in Section 4.1.2.2 [Processing options], page 159.
11.3.22 Statistical operations (statistics.h)

After reading a dataset into memory from a file or fully simulating it with another process, the most common processes that will be done on it are statistical operations to let you quantify different aspects of the data. The functions in this section describe Gnuastro’s current set of tools for this job. All these functions can work on any numeric data type natively (see Section 4.5 [Numeric data types], page 178) and can also work on tiles over a dataset. Hence the inputs and outputs are in Gnuastro’s Section 11.3.6.1 [Generic data container (gal_data_t)], page 542.

GAL_STATISTICS_SIG_CLIP_MAX_CONVERGE [Macro]
The maximum number of clips, when $\sigma$-clipping should be done by convergence. If the clipping does not converge before making this many clips, all $\sigma$-clipping outputs will be NaN.

GAL_STATISTICS_MODE_GOOD_SYM [Macro]
The minimum acceptable symmetricity of the mode calculation. If the symmetricity of the derived mode is less than this value, all the returned values by gal_statistics_mode will have a value of NaN.

GAL_STATISTICS_BINS_INVALID [Macro]
GAL_STATISTICS_BINS_REGULAR [Macro]
GAL_STATISTICS_BINS_IRREGULAR [Macro]
Macros used to identify if the regularity of the bins when defining bins.

gal_data_t * [Function]
gal_statistics_number (gal_data_t *input)
Return a single-element dataset with type size_t which contains the number of non-blank elements in input.

gal_data_t * [Function]
gal_statistics_minimum (gal_data_t *input)
Return a single-element dataset containing the minimum non-blank value in input. The numerical datatype of the output is the same as input.

gal_data_t * [Function]
gal_statistics_maximum (gal_data_t *input)
Return a single-element dataset containing the maximum non-blank value in input. The numerical datatype of the output is the same as input.

gal_data_t * [Function]
gal_statistics_sum (gal_data_t *input)
Return a single-element (double or float64) dataset containing the sum of the non-blank values in input.

gal_data_t * [Function]
gal_statistics_mean (gal_data_t *input)
Return a single-element (double or float64) dataset containing the mean of the non-blank values in input.
### Chapter 11: Library

#### gal_data_t *

**gal_statistics_std (gal_data_t *input)**

Return a single-element (double or float64) dataset containing the standard deviation of the non-blank values in input.

#### gal_data_t *

**gal_statistics_mean_std (gal_data_t *input)**

Return a two-element (double or float64) dataset containing the mean and standard deviation of the non-blank values in input. The first element of the returned dataset is the mean and the second is the standard deviation.

This function will calculate both values in one pass over the dataset. Hence when both the mean and standard deviation of a dataset are necessary, this function is much more efficient than calling `gal_statistics_mean` and `gal_statistics_std` separately.

#### size_t

**gal_statistics_quantile_index (size_t size, double quantile)**

Return the index of the element that has a quantile of quantile assuming the dataset has size elements.

#### gal_data_t *

**gal_statistics_quantile (gal_data_t *input, double quantile, int inplace)**

Return a single-element dataset containing the value with in a quantile quantile of the non-blank values in input. The numerical datatyp of the output is the same as input. See `gal_statistics_median` for a description of inplace.

#### size_t

**gal_statistics_quantile_function_index (gal_data_t *input, gal_data_t *value, int inplace)**

Return the index of the quantile function (inverse quantile) of input at value. In other words, this function will return the index of the nearest element (of a sorted and non-blank) input to value. If the value is outside the range of the input, then this function will return GAL_BLANK_SIZE_T.

#### gal_data_t *

**gal_statistics_quantile_function (gal_data_t *input, gal_data_t *value, int inplace)**

Return a single-element dataset containing the quantile function of the non-blank values in input at value (a single-element dataset). The numerical data type is
The returned dataset is float64 (or double). In other words, this function will return the quantile of value in input. value has to have the same type as input. See gal_statistics_median for a description of inplace.

When all elements are blank, the returned value will be NaN. If the value is smaller than the input’s smallest element, the returned value will be negative infinity. If the value is larger than the input’s largest element, then the returned value will be positive infinity.

```c
gal_data_t *
gal_statistics_unique (gal_data_t *input, int inplace)
```

Return a 1D dataset with the same numeric data type as the input, but only containing its unique elements and without any (possible) blank/NaN elements. Note that the input’s number of dimensions is irrelevant for this function. If inplace is not zero, then the unique values will over-write the allocated space of the input, otherwise a new space will be allocated and the input will not be touched.

```c
gal_data_t *
gal_statistics_mode (gal_data_t *input, float mirrordist, int inplace)
```

Return a four-element (double or float64) dataset that contains the mode of the input distribution. This function implements the non-parametric algorithm to find the mode that is described in Appendix C of Akhlaghi and Ichikawa [2015] (https://arxiv.org/abs/1505.01664).

In short it compares the actual distribution and its “mirror distribution” to find the mode. In order to be efficient, you can determine how far the comparison goes away from the mirror through the mirrordist parameter (think of it as a multiple of sigma/error). See gal_statistics_median for a description of inplace.

The output array has the following elements (in the given order, note that counting in C starts from 0).

- array[0]: mode
- array[1]: mode quantile.
- array[2]: symmetricity.
- array[3]: value at the end of symmetricity.

```c
gal_data_t *
gal_statistics_mode_mirror_plots (gal_data_t *input, gal_data_t *value,
    size_t numbins, int inplace, double *mirror_val)
```

Make a mirrored histogram and cumulative frequency plot (with numbins) with the mirror distribution of the input having a value in value. If all the input elements are blank, or the mirror value is outside the range of the input, this function will return a NULL pointer.

The output is a list of data structures (see Section 11.3.8.9 [List of gal_data_t], page 566): the first is the bins with one bin at the mirror point, the second is the histogram with a maximum of one and the third is the cumulative frequency plot (with a maximum of one).
int

gal_statistics_is_sorted (gal_data_t *input, int updateflags)

Return 0 if the input is not sorted, if it is sorted, this function will return 1 and 2 if it is increasing or decreasing, respectively. This function will abort with an error if input has zero elements and will return 1 (sorted, increasing) when there is only one element. This function will only look into the dataset if the GAL_DATA_FLAG_SORT_CH bit of input->flag is 0, see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542.

When the flags don’t indicate a previous check and updateflags is non-zero, this function will set the flags appropriately to avoid having to re-check the dataset in future calls (this can be very useful when repeated checks are necessary). When updateflags==0, this function has no side-effects on the dataset: it will not toggle the flags.

If you want to re-check a dataset with the blank-value-check flag already set (for example if you have made changes to it), then explicitly set the GAL_DATA_FLAG_SORT_CH bit to zero before calling this function. When there are no other flags, you can simply set the flags to zero (with input->flag=0), otherwise you can use this expression:

        input->flag &= ~GAL_DATA_FLAG_SORT_CH;

void

gal_statistics_sort_increasing (gal_data_t *input)

Sort the input dataset (in place) in an increasing order and toggle the sort-related bit flags accordingly.

void

gal_statistics_sort_decreasing (gal_data_t *input)

Sort the input dataset (in place) in a decreasing order and toggle the sort-related bit flags accordingly.

gal_data_t *

gal_statistics_no_blank_sorted (gal_data_t *input, int inplace)

Remove all the blanks and sort the input dataset. If inplace is non-zero this will happen on the input dataset (in the allocated space of the input dataset). However, if inplace is zero, this function will allocate a new copy of the dataset and work on that. Therefore if inplace==0, the input dataset will be modified.

This function uses the bit flags of the input, so if you have modified the dataset, set input->flag=0 before calling this function. Also note that inplace is only for the dataset elements. Therefore even when inplace==0, if the input is already sorted and has no blank values, then the flags will be updated to show this.

If all the elements were blank, then the returned dataset’s size will be zero. This is thus a good parameter to check after calling this function to see if there actually were any non-blank elements in the input or not and take the appropriate measure. This can help avoid strange bugs in later steps. The flags of a zero-sized returned dataset will indicate that it has no blanks and is sorted in an increasing order. Even if having blank values or being sorted is not defined on a zero-element dataset, it is up to the
caller to choose what they will do with a zero-element dataset. The flags have to be set after this function any way.

\[
\text{gal_data_t * } \text{gal_statistics_regular_bins (gal.data_t *input, gal.data_t *inrange, size_t numbins, double onebinstart)}
\]

Generate an array of regularly spaced elements as a 1D array (column) of type double (i.e., float64, it has to be double to account for small differences on the bin edges). The input arguments are described below

**input**  The dataset you want to apply the bins to. This is only necessary if the range argument is not complete, see below. If inrange has all the necessary information, you can pass a NULL pointer for this.

**inrange**  This dataset keeps the desired range along each dimension of the input data structure, it has to be in float (i.e., float32) type.

- If you want the full range of the dataset (in any dimensions, then just set inrange to NULL and the range will be specified from the minimum and maximum value of the dataset (input cannot be NULL in this case).
- If there is one element for each dimension in range, then it is viewed as a quantile (Q), and the range will be: ‘Q to 1-Q’.
- If there are two elements for each dimension in range, then they are assumed to be your desired minimum and maximum values. When either of the two are NaN, the minimum and maximum will be calculated for it.

**numbins**  The number of bins: must be larger than 0.

**onebinstart**  A desired value to start one bin. Note that with this option, the bins won’t start and end exactly on the given range values, it will be slightly shifted to accommodate this request (enough for the bin containing the value to start at it). If you don’t have any preference on where to start a bin, set this to NaN.

\[
\text{gal_data_t * } \text{gal_statistics_histogram (gal.data_t *input, gal.data_t *bins, int normalize, int maxone)}
\]

Make a histogram of all the elements in the given dataset with bin values that are defined in the bins structure (see gal_statistics_regular_bins, they currently have to be equally spaced). The returned histogram is a 1-D gal.data_t of type GAL_TYPE_FLOAT32, with the same number of elements as bins. For each bin, it will contain the number of input elements that fell inside of that bin.

Let’s write the center of the \(i\)th element of the bin array as \(b_i\), and the fixed half-bin width as \(h\). Then element \(j\) of the input array \((in_j)\) will be counted in \(b_i\) if \((b_i - h) \leq in_j < (b_i + h)\). However, if \(in_j\) is somewhere in the last bin, the condition changes to \((b_i - h) \leq in_j \leq (b_i + h)\).
If normalize!=0, the histogram will be “normalized” such that the sum of the counts column will be one. In other words, all the counts in every bin will be divided by the total number of counts. If maxone!=0, the histogram’s maximum count will be 1. In other words, the counts in every bin will be divided by the value of the maximum. In both of these cases, the output dataset will have a GAL_DATA_FLOAT32 datatype.

```
[Function]
gal_data_t *
gal_statistics_histogram2d (gal_data_t *input, gal_data_t *bins)
```

This function is very similar to `gal_statistics_histogram`, but will build a 2D histogram (count how many of the elements of `input` are within a 2D box. The bins comprising the first dimension of the 2D box are defined by `bins`. The bins of the second dimension are defined by `bins->next` (`bins` is a Section 11.3.8.9 [List of `gal_data_t`], page 566). Both the `bin` and `bin->next` can be created with `gal_statistics_regular_bins`.

This function returns a list of `gal_data_t` with three nodes/columns, so you can directly write them into a table (see Section 11.3.10 [Table input output (table.h)], page 568). Assuming `bins` has $N_1$ bins and `bins->next` has $N_2$ bins, each node/column of the returned output is a 1D array with $N_1 \times N_2$ elements. The first and second columns are the center of the 2D bin along the first and second dimensions and have a `double` data type. The third column is the 2D histogram (the number of input elements that have a value within that 2D bin) and has a `uint32` data type (see Section 4.5 [Numeric data types], page 178).

```
[Function]
gal_data_t *
gal_statistics_cfp (gal_data_t *input, gal_data_t *bins, int normalize)
```

Make a cumulative frequency plot (CFP) of all the elements in `input` with bin values that are defined in the `bins` structure (see `gal_statistics_regular_bins`).

The CFP is built from the histogram: in each bin, the value is the sum of all previous bins in the histogram. Thus, if you have already calculated the histogram before calling this function, you can pass it onto this function as the data structure in `bins->next` (see List of `gal_data_t`). If `bin->next!=NULL`, then it is assumed to be the histogram. If it is `NULL`, then the histogram will be calculated internally and freed after the job is finished.

When a histogram is given and it is normalized, the CFP will also be normalized (even if the normalized flag is not set here): note that a normalized CFP’s maximum value is 1.

```
[Function]
gal_data_t *
gal_statistics_sigma_clip (gal_data_t *input, float multip, float param, int inplace, int quiet)
```

Apply $\sigma$-clipping on a given dataset and return a dataset that contains the results. For a description of $\sigma$-clipping see Section 7.1.3 [Sigma clipping], page 343. `multip` is the multiple of the standard deviation (or $\sigma$, that is used to define outliers in each round of clipping).

The role of `param` is determined based on its value. If `param` is larger than 1 (one), it must be an integer and will be interpreted as the number clips to do. If it is less than 1 (one), it is interpreted as the tolerance level to stop the iteration.
The returned dataset (let’s call it `out`) contains a four-element array with type `GAL_TYPE_FLOAT32`. The final number of clips is stored in the `out->status`.

```
float *array=out->array;
array[0]: Number of points used.
array[1]: Median.
array[2]: Mean.
array[3]: Standard deviation.
```

If the $\sigma$-clipping doesn’t converge or all input elements are blank, then this function will return NaN values for all the elements above.

```c
[Function]
gal_data_t * gal_statistics_outlier_bydistance (int pos1_neg0, gal_data_t *input, size_t window_size, float sigma, float sigclip_multip, float sigclip_param, int inplace, int quiet)
```

Find the first positive outlier (if `pos1_neg0!=0`) in the `input` distribution. When `pos1_neg0==0`, the same algorithm goes to the start of the dataset. The returned dataset contains a single element: the first positive outlier. It is one of the dataset’s elements, in the same type as the input. If the process fails for any reason (for example no outlier was found), a NULL pointer will be returned.

All (possibly existing) blank elements are first removed from the input dataset, then it is sorted. A sliding window of `window_size` elements is parsed over the dataset. Starting from the `window_size`-th element of the dataset, in the direction of increasing values. This window is used as a reference. The first element where the distance to the previous (sorted) element is $\sigma$ units away from the distribution of distances in its window is considered an outlier and returned by this function.

Formally, if we assume there are $N$ non-blank elements. They are first sorted. Searching for the outlier starts on element $W$. Let’s take $v_i$ to be the $i$-th element of the sorted input (with no blank values) and $m$ and $\sigma$ as the $\sigma$-clipped median and standard deviation from the distances of the previous $W$ elements (not including $v_i$). If the value given to `sigma` is displayed with $s$, the $i$-th element is considered as an outlier when the condition below is true.

\[
\frac{(v_i - v_{i-1}) - m}{\sigma} > s
\]

The `sigclip_multip` and `sigclip_param` arguments specify the properties of the $\sigma$-clipping (see Section 7.1.3 [Sigma clipping], page 343, for more). You see that by this definition, the outlier cannot be any of the lower half elements. The advantage of this algorithm compared to $\sigma$-clipping is that it only looks backwards (in the sorted array) and parses it in one direction.

If `inplace!=0`, the removing of blank elements and sorting will be done within the input dataset’s allocated space. Otherwise, this function will internally allocate (and later free) the necessary space to keep the intermediate space that this process requires.

If `quiet!=0`, this function will report the parameters every time it moves the window as a separate line with several columns. The first column is the value, the second (in
square brackets) is the sorted index, the third is the distance of this element from the previous one. The Fourth and fifth (in parenthesis) are the median and standard deviation of the \( \sigma \)-clipped distribution within the window and the last column is the difference between the third and fourth, divided by the fifth.

\[
\text{gal_data_t} * \\
gal\text{-statistics}_{\text{outlier flat cfp}} (\text{gal}\_\text{data_t}*\text{input}, \text{size_t numprev}, \text{float sigclip\_multip}, \text{float sigclip\_param}, \text{float thresh}, \text{size_t numcontig}, \\
\text{int inplace}, \text{int quiet}, \text{size_t *index})
\]

Return the first element in the given dataset where the cumulative frequency plot first becomes significantly flat for a sufficient number of elements. The returned dataset only has one element (with the same type as the input). If \text{index}!=\text{NULL}, the index (counting from zero, after sorting the dataset and removing any blanks) is written in the space that \text{index} points to. If no sufficiently flat portion is found, the returned pointer will be \text{NULL}.

The flatness on the cumulative frequency plot is defined like this (see Section 7.1.1 [Histogram and Cumulative Frequency Plot], page 337): on the sorted dataset, for every point \( a_i \), we calculate \( d_i = a_{i+2} - a_{i-2} \). This done on the first \( N \) elements (value of \text{numprev}). After element \( a_{N+2} \), we start estimating the flatness as follows: for every element we use the \( N \), \( d_i \) measurements before it as the reference. Let’s call this set \( D_i \) for element \( i \). The \( \sigma \)-clipped median \( (m) \) and standard deviation \( (s) \) of \( D_i \) are then calculated. The \( \sigma \)-clipping can be configured with the two \text{sigclip\_param} and \text{sigclip\_multip} arguments.

Taking \( t \) as the significance threshold (value to \text{thresh}), a point is considered flat when \( a_i > m + t\sigma \). But a single point satisfying this condition will probably just be due to noise. To make a more robust estimate, this significance/condition has to hold for \text{numcontig} contiguous elements after \( a_i \). When this is satisfied, \( a_i \) is returned as the point where the distribution’s cumulative frequency plot becomes flat.

To get a good estimate of \( m \) and \( s \), it is thus recommended to set \text{numprev} as large as possible. However, be careful not to set it too high: the checks in the paragraph above are not done on the first \text{numprev} elements and this function assumes the flatness occurs after them. Also, be sure that the value to \text{numcontig} is much less than \text{numprev}, otherwise \( \sigma \)-clipping may not be able to remove the immediate outliers in \( D_i \) near the boundary of the flat region.

When \text{quiet}==0, the basic measurements done on each element are printed on the command-line (good for finding the best parameters). When \text{inplace}!=0, the sorting and removal of blank elements is done on the input dataset, so the input may be altered after this function.

11.3.23 Binary datasets (binary.h)

Binary datasets only have two (usable) values: 0 (also known as background) or 1 (also known as foreground). They are created after some binary classification is applied to the dataset. The most common is thresholding: for example in an image, pixels with a value above the threshold are given a value of 1 and those with a value less than the threshold are assigned a value of 0.
Since there is only two values, in the processing of binary images, you are usually concerned with the positioning of an element and its vicinity (neighbors). When a dataset has more than one dimension, multiple classes of immediate neighbors (that are touching the element) can be defined for each data-element. To separate these different classes of immediate neighbors, we define _connectivity_.

The classification is done by the distance from element center to the neighbor's center. The nearest immediate neighbors have a connectivity of 1, the second nearest class of neighbors have a connectivity of 2 and so on. In total, the largest possible connectivity for data with \( n_{\text{dim}} \) dimensions is \( n_{\text{dim}} \). For example in a 2D dataset, 4-connected neighbors (that share an edge and have a distance of 1 pixel) have a connectivity of 1. The other 4 neighbors that only share a vertice (with a distance of \( \sqrt{2} \) pixels) have a connectivity of 2. Conventionally, the class of connectivity-2 neighbors also includes the connectivity 1 neighbors, so for example we call them 8-connected neighbors in 2D datasets.

Ideally, one bit is sufficient for each element of a binary dataset. However, CPUs are not designed to work on individual bits, the smallest unit of memory addresses is a byte (containing 8 bits on modern CPUs). Therefore, in Gnuastro, the type used for binary dataset is \texttt{uint8_t} (see Section 4.5 [Numeric data types], page 178). Although it does take 8-times more memory, this choice offers much better performance and the some extra (useful) features.

The advantage of using a full byte for each element of a binary dataset is that you can also have other values (that will be ignored in the processing). One such common “other” value in real datasets is a blank value (to mark regions that should not be processed because there is no data). The constant \texttt{GAL_BLANK_UINT8} value must be used in these cases (see Section 11.3.5 [Library blank values (\texttt{blank.h})], page 538). Another is some temporary value(s) that can be given to a processed pixel to avoid having another copy of the dataset as in \texttt{GAL_BINARY_TMP_VALUE} that is described below.

\texttt{GAL_BINARY_TMP_VALUE}  
[Macro]  
The functions described below work on a \texttt{uint8_t} type dataset with values of 1 or 0 (no other pixel will be touched). However, in some cases, it is necessary to put temporary values in each element during the processing of the functions. This temporary value has a special meaning for the operation and will be operated on. So if your input datasets have values other than 0 and 1 that you don’t want these functions to work on, be sure they are not equal to this macro’s value. Note that this value is also different from \texttt{GAL_BLANK_UINT8}, so your input datasets may also contain blank elements.

\texttt{gal_data_t *} \hspace{1cm} \texttt{gal_binary_erode (gal_data_t *input, size_t num, int connectivity, int inplace)}  
[Function]  
Do \texttt{num} erosions on the \texttt{connectivity}-connected neighbors of \texttt{input} (see above for the definition of connectivity).  
If \texttt{inplace} is non-zero \texttt{and} the input’s type is \texttt{GAL_TYPE_UINT8}, then the erosion will be done within the input dataset and the returned pointer will be \texttt{input}. Otherwise, \texttt{input} is copied (and converted if necessary) to \texttt{GAL_TYPE_UINT8} and erosion will be done on this new dataset which will also be returned. This function will only work on the elements with a value of 1 or 0. It will leave all the rest unchanged.
Erosion (inverse of dilation) is an operation in mathematical morphology where each foreground pixel that is touching a background pixel is flipped (changed to background). The connectivity value determines the definition of “touching”. Erosion will thus decrease the area of the foreground regions by one layer of pixels.

```c
gal_data_t * gal_binary_dilate (gal_data_t *input, size_t num, int connectivity, int inplace)
```

Do `num` dilations on the `connectivity`-connected neighbors of `input` (see above for the definition of connectivity). For more on `inplace` and the output, see `gal_binary_erode`.

Dilation (inverse of erosion) is an operation in mathematical morphology where each background pixel that is touching a foreground pixel is flipped (changed to foreground). The connectivity value determines the definition of “touching”. Dilation will thus increase the area of the foreground regions by one layer of pixels.

```c
gal_data_t * gal_binary_open (gal_data_t *input, size_t num, int connectivity, int inplace)
```

Do `num` openings on the `connectivity`-connected neighbors of `input` (see above for the definition of connectivity). For more on `inplace` and the output, see `gal_binary_erode`.

Opening is an operation in mathematical morphology which is defined as erosion followed by dilation (see above for the definitions of erosion and dilation). Opening will thus remove the outer structure of the foreground. In this implementation, `num` erosions are going to be applied on the dataset, then `num` dilations.

```c
size_t gal_binary_connected_components (gal_data_t *binary, gal_data_t **out, int connectivity)
```

Return the number of connected components in `binary` through the breadth first search algorithm (finding all pixels belonging to one component before going on to the next). Connection between two pixels is defined based on the value to `connectivity`. `out` is a dataset with the same size as `binary` with `GAL_TYPE_INT32` type. Every pixel in `out` will have the label of the connected component it belongs to. The labeling of connected components starts from 1, so a label of zero is given to the input’s background pixels.

When `*out!=NULL` (its space is already allocated), it will be cleared (to zero) at the start of this function. Otherwise, when `*out==NULL`, the necessary dataset to keep the output will be allocated by this function.

`binary` must have a type of `GAL_TYPE_UINT8`, otherwise this function will abort with an error. Other than blank pixels (with a value of `GAL_BLANK_UINT8` defined in Section 11.3.5 [Library blank values (blank.h)], page 538), all other non-zero pixels in `binary` will be considered as foreground (and will be labeled). Blank pixels in the input will also be blank in the output.
gal_data_t *
gal_binary_connected_indices (gal_data_t *binary, int connectivity)

Build a gal_data_t linked list, where each node of the list contains an array with indices of the connected regions. Therefore the arrays of each node can have a different size. Note that the indices will only be calculated on the pixels with a value of 1 and internally, it will temporarily change the values to 2 (and return them back to 1 in the end).

gal_data_t *
gal_binary_connected_adjacency_matrix (gal_data_t *adjacency, size_t *numconnected)

Find the number of connected labels and new labels based on an adjacency matrix, which must be a square binary array (type GAL_TYPE_UINT8). The returned dataset is a list of new labels for each old label. In other words, this function will find the objects that are connected (possibly through a third object) and in the output array, the respective elements for all input labels is going to have the same value. The total number of connected labels is put into the space that numconnected points to.

An adjacency matrix defines connection between two labels. For example, let’s assume we have 5 labels and we know that labels 1 and 5 are connected to label 3, but are not connected with each other. Also, labels 2 and 4 are not touching any other label. So in total we have 3 final labels: one combined object (merged from labels 1, 3, and 5) and the initial labels 2 and 4. The input adjacency matrix would look like this (note the extra row and column for a label 0 which is ignored):

<table>
<thead>
<tr>
<th>INPUT</th>
<th>OUTPUT</th>
</tr>
</thead>
<tbody>
<tr>
<td>in_lab 1 2 3 4 5</td>
<td>numconnected = 3</td>
</tr>
<tr>
<td>0 0 0 0 0 0</td>
<td></td>
</tr>
<tr>
<td>in_lab 1 --&gt; 0 0 0 1 0 0</td>
<td>Returned: new labels for the</td>
</tr>
<tr>
<td>in_lab 2 --&gt; 0 0 0 0 0</td>
<td>5 initial objects</td>
</tr>
<tr>
<td>in_lab 3 --&gt; 0 1 0 0 0 1</td>
<td></td>
</tr>
<tr>
<td>in_lab 4 --&gt; 0 0 0 0 0</td>
<td>0</td>
</tr>
<tr>
<td>in_lab 5 --&gt; 0 0 0 1 0 0</td>
<td></td>
</tr>
</tbody>
</table>

Although the adjacency matrix as used here is symmetric, currently this function assumes that it is filled on both sides of the diagonal.

gal_data_t *
gal_binary_connected_adjacency_list (gal_list_sizet **listarr, size_t number, size_t minmmapsize, int quietmmap, size_t *numconnected)

Find the number of connected labels and new labels based on an adjacency list. The output of this function is identical to that of gal_binary_connected_adjacency_matrix. But the major difference is that it uses a list of connected labels to each label instead of a square adjacency matrix. This is done because when the number of labels becomes very large (for example on the scale of 100,000), the adjacency matrix can consume more than 10GB of RAM!

The input list has the following format: it is an array of pointers to gal_list_sizet t * (or gal_list_sizet t **). The array has number elements and each listarr[i]
is a linked list of `gal_list_sizet_t *`. As a demonstration, the input of the same example in `gal_binary_connected_adjacency_matrix` would look like below and the output of this function will be identical to there.

```c
listarr[0] = NULL
listarr[1] = 3
listarr[2] = NULL
listarr[4] = NULL
listarr[5] = 3
```

From this example, it is already clear that this method will consume far less memory. But because it needs to parse lists (and not easily jump between array elements), it can be slower. But in scenarios where there are too many objects (that may exceed the whole system's RAM+SWAP), this option is a good alternative and the drop in processing speed is worth getting the job done.

Similar to `gal_binary_connected_adjacency_matrix`, this function will write the final number of connected labels in `numconnected`. But since it takes no `gal_data_t *` argument (where it can inherit the `minmapsize` and `quietmmap` parameters), it also needs these as input. For more on `minmapsize` and `quietmmap`, see Section 4.6 [Memory management], page 180.

**gal_data_t * [Function]**

```c
gal_binary_holes_label (gal_data_t *input, int connectivity, size_t *numholes)
```

Label all the holes in the foreground (non-zero elements in input) as independent regions. Holes are background regions (zero-valued in input) that are fully surrounded by the foreground, as defined by `connectivity`. The returned dataset has a 32-bit signed integer type with the size of the input. All holes in the input will have labels/counters greater or equal to 1. The rest of the background regions will still have a value of 0 and the initial foreground pixels will have a value of −1. The total number of holes will be written where `numholes` points to.

**void [Function]**

```c
gal_binary_holes_fill (gal_data_t *input, int connectivity, size_t maxsize)
```

Fill all the holes (0 valued pixels surrounded by 1 valued pixels) of the binary input dataset. The connectivity of the holes can be set with `connectivity`. Holes larger than `maxsize` are not filled. This function currently only works on a 2D dataset.

### 11.3.24 Labeled datasets (`label.h`)

A labeled dataset is one where each element/pixel has an integer label (or counter). The label identifies the group/class that the element belongs to. This form of labeling allows the higher-level study of all pixels within a certain class.

For example, to detect objects/targets in an image/dataset, you can apply a threshold to separate the noise from the signal (to detect diffuse signal, a threshold is useless and more advanced methods are necessary, for example Section 7.2 [NoiseChisel], page 362). But the output of detection is a binary dataset (which is just a very low-level labeling of 0 for noise and 1 for signal).
The raw detection map is therefore hardly useful for any kind of analysis on objects/targets in the image. One solution is to use a connected-components algorithm (see `gal_binary_connected_components` in Section 11.3.23 [Binary datasets (binary.h)], page 640). It is a simple and useful way to separate/label connected patches in the foreground. This higher-level (but still elementary) labeling therefore allows you to count how many connected patches of signal there are in the dataset and is a major improvement compared to the raw detection.

However, when your objects/targets are touching, the simple connected components algorithm is not enough and a still higher-level labeling mechanism is necessary. This brings us to the necessity of the functions in this part of Gnuastro’s library. The main inputs to the functions in this section are already labeled datasets (for example with the connected components algorithm above).

Each of the labeled regions are independent of each other (the labels specify different classes of targets). Therefore, especially in large datasets, it is often useful to process each label on independent CPU threads in parallel rather than in series. Therefore the functions of this section actually use an array of pixel/element indices (belonging to each label/class) as the main identifier of a region. Using indices will also allow processing of overlapping labels (for example in deblending problems). Just note that overlapping labels are not yet implemented, but planned. You can use `gal_label_indexs` to generate lists of indices belonging to separate classes from the labeled input.

```c
GAL_LABEL_INIT
GAL_LABEL_RIVER
GAL_LABEL_TMPCHECK
```

Special negative integer values used internally by some of the functions in this section. Recall that meaningful labels are considered to be positive integers (≥ 1). Zero is conventionally kept for regions with no labels, therefore negative integers can be used for any extra classification in the labeled datasets.

```c
gal_data_t *
gal_label_indexs (gal_data_t *labels, size_t numlabs, size_t minmapsize, int quietmmap)
```

Return an array of `gal_data_t` containers, each containing the pixel indices of the respective label (see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542). `labels` contains the label of each element and has to have an `GAL_TYPE_INT32` type (see Section 11.3.3 [Library data types (type.h)], page 532). Only positive (greater than zero) values in `labels` will be used/indexed, other elements will be ignored.

Meaningful labels start from 1 and not 0, therefore the output array of `gal_data_t` will contain `numlabs+1` elements. The first (zero-th) element of the output (indices[0] in the example below) will be initialized to a dataset with zero elements. This will allow easy (non-confusing) access to the indices of each (meaningful) label.

`numlabs` is the number of labels in the dataset. If it is given a value of zero, then the maximum value in the input (largest label) will be found and used. Therefore if it is given, but smaller than the actual number of labels, this function may/will crash (it will write in unallocated space). `numlabs` is therefore useful in a highly optimized/checked environment.
For example, if the returned array is called `indexes`, then `indexes[10].size` contains the number of elements that have a label of 10 in `labels` and `indexes[10].array` is an array (after casting to `size_t *`) containing the indices of each one of those elements/pixels.

By `index` we mean the 1D position: the input number of dimensions is irrelevant (any dimensionality is supported). In other words, each element’s index is the number of elements/pixels between it and the dataset’s first element/pixel. Therefore it is always greater or equal to zero and stored in `size_t` type.

**size_t**

`gal_label_watershed` (*gal_data_t* `values`, *gal_data_t* `indexes`, *gal_data_t* `label`, *size_t* `topinds`, *int* `min0_max1`)

Use the watershed algorithm to “over-segment” the pixels in the `indexes` dataset based on values in the `values` dataset. Internally, each local extrema (maximum or minimum, based on `min0_max1`) and its surrounding pixels will be given a unique label. For demonstration, see Figures 8 and 9 of Akhlaghi and Ichikawa [2015] (http://arxiv.org/abs/1505.01664). If `topinds`!=NULL, it is assumed to point to an already allocated space to write the index of each clump’s local extrema, otherwise, it is ignored.

The `values` dataset must have a 32-bit floating point type (GAL_TYPE_FLOAT32, see Section 11.3.3 [Library data types (type.h)], page 532) and will only be read by this function. `indexes` must contain the indices of the elements/pixels that will be over-segmented by this function and have a GAL_TYPE_SIZE_T type, see the description of `gal_label_indexs`, above. The final labels will be written in the respective positions of `labels`, which must have a GAL_TYPE_INT32 type and be the same size as `values`.

When `indexes` is already sorted, this function will ignore `min0_max1`. To judge if the dataset is sorted or not (by the values the indices correspond to in `values`, not the actual indices), this function will look into the bits of `indexes->flag`, for the respective bit flags, see Section 11.3.6.1 [Generic data container (gal_data_t)], page 542. If `indexes` is not already sorted, this function will sort it according to the values of the respective pixel in `values`. The increasing/decreasing order will be determined by `min0_max1`. Note that if this function is called on multiple threads and `values` points to a different array on each thread, this function will not return a reasonable result. In this case, please sort `indexes` prior to calling this function (see `gal_qsort_index_multi_d` in Section 11.3.18 [Qsort functions (qsort.h)], page 623).

When `indexes` is decreasing (increasing), or `min0_max1` is 1 (0), local minima (maxima), are considered rivers (watersheds) and given a label of GAL_LABEL_RIVER (see above).

Note that rivers/watersheds will also be formed on the edges of the labeled regions or when the labeled pixels touch a blank pixel. Therefore this function will need to check for the presence of blank values. To be most efficient, it is thus recommended to use `gal_blank_present` (with `updateflag`=1) prior to calling this function (see

---

26 The watershed algorithm was initially introduced by Vincent and Soille (https://doi.org/10.1109/34.87344). It starts from the minima and puts the pixels in, one by one, to grow them until the touch (create a watershed). For more, also see the Wikipedia article: https://en.wikipedia.org/wiki/Watershed_%28image_processing%29.
Section 11.3.5 [Library blank values (blank.h)], page 538. Once the flag has been set, no other function (including this one) that needs special behavior for blank pixels will have to parse the dataset to see if it has blank values any more.

If you are sure your dataset doesn’t have blank values (by the design of your software), to avoid an extra parsing of the dataset and improve performance, you can set the two bits manually (see the description of flags in Section 11.3.6.1 [Generic data container (gal_data_t)], page 542):

```c
input->flag |= GAL_DATA_FLAG_BLANK_CH; /* Set bit to 1. */
input->flag &= ~GAL_DATA_FLAG_HASBLANK; /* Set bit to 0. */
```

**void**

`gal_label_clump_significance (gal_data_t *values, gal_data_t *std, gal_data_t *label, gal_data_t *indexes, struct gal_tile_two_layer_params *tl, size_t numclumps, size_t minarea, int variance, int keepsmall, gal_data_t *sig, gal_data_t *sigind)`

This function is usually called after `gal_label_watershed`, and is used as a measure to identify which over-segmented “clumps” are real and which are noise.

A measurement is done on each clump (using the `values` and `std` datasets, see below). To help in multi-threaded environments, the operation is only done on pixels which are indexed in `indexes`. It is expected for `indexes` to be sorted by their values in `values`. If not sorted, the measurement may not be reliable. If sorted in a decreasing order, then clump building will start from their highest value and vice-versa. See the description of `gal_label_watershed` for more on `indexes`.

Each “clump” (identified by a positive integer) is assumed to be surrounded by at least one river/watershed pixel (with a non-positive label). This function will parse the pixels identified in `indexes` and make a measurement on each clump and over all the river/watershed pixels. The number of clumps (`numclumps`) must be given as an input argument and any clump that is smaller than `minarea` is ignored (because of scatter). If `variance` is non-zero, then the `std` dataset is interpreted as variance, not standard deviation.

The `values` and `std` datasets must have a `float` (32-bit floating point) type. Also, `label` and `indexes` must respectively have `int32` and `size_t` types. `values` and `label` must have the same size, but `std` can have three possible sizes: 1) a single element (which will be used for the whole dataset, 2) the same size as `values` (so a different error can be assigned to every pixel), 3) a single value for each tile, based on the `tl` tessellation (see Section 11.3.15.2 [Tile grid], page 614). In the last case, a tile/value will be associated to each clump based on its flux-weighted (only positive values) center.

The main output is an internally allocated, 1-dimensional array with one value per label. The array information (length, type, etc) will be written into the `sig` generic data container. Therefore `sig->array` must be NULL when this function is called. After this function, the details of the array (number of elements, type and size, etc) will be written in to the various components of `sig`, see the definition of `gal_data_t` in Section 11.3.6.1 [Generic data container (gal_data_t)], page 542. Therefore `sig` must already be allocated before calling this function.
Optionally (when sigind!=NULL, similar to sig) the clump labels of each measurement in sig will be written in sigind->array. If keepsmall zero, small clumps (where no measurement is made) will not be included in the output table.

This function is initially intended for a multi-threaded environment. In such cases, you will be writing arrays of clump measures from different regions in parallel into an array of gal_data_t. You can simply allocate (and initialize), such an array with the gal_data_arraycalloc function in Section 11.3.6.3 [Arrays of datasets], page 548. For example if the gal_data_t array is called array, you can pass &array[i] as sig.

Along with some other functions in label.h, this function was initially written for Section 7.3 [Segment], page 381. The description of the parameter used to measure a clump’s significance is fully given in Akhlaghi [2019] (https://arxiv.org/abs/1909.11230).

### [Function]

void gal_label_grow_indices (gal_data_t *labels, gal_data_t *indexs, int withrivers, int connectivity)

Grow the (positive) labels of labels over the pixels in indexs (see description of gal_label_indices). The pixels (position in indexs, values in labels) that must be “grown” must have a value of GAL_LABEL_INIT in labels before calling this function. For a demonstration see Columns 2 and 3 of Figure 10 in Akhlaghi and Ichikawa [2015] (http://arxiv.org/abs/1505.01664).

In many aspects, this function is very similar to over-segmentation (watershed algorithm, gal_label_watershed). The big difference is that in over-segmentation local maximums (that aren’t touching any alreadylabeled pixel) get a separate label. However, here the final number of labels will not change. All pixels that aren’t directly touching a labeled pixel just get pushed back to the start of the loop, and the loop iterates until its size doesn’t change any more. This is because in a generic scenario some of the indexed pixels might not be reachable through other indexed pixels.

The next major difference with over-segmentation is that when there is only one label in growth region(s), it is not mandatory for indexs to be sorted by values. If there are multiple labeled regions in growth region(s), then values are important and you can use qsort with gal_qsort_index_single_d to sort the indices by values in a separate array (see Section 11.3.18 [Qsort functions (qsort.h)], page 623).

This function looks for positive-valued neighbors of each pixel in indexs and will label a pixel if it touches one. Therefore, it is very important that only pixels/labels that are intended for growth have positive values in labels before calling this function. Any non-positive (zero or negative) value will be ignored as a label by this function. Thus, it is recommended that while filling in the indexs array values, you initialize all the pixels that are in indexs with GAL_LABEL_INIT, and set non-labeled pixels that you don’t want to grow to 0.

This function will write into both the input datasets. After this function, some of the non-positive labels pixels will have a new positivelabel and the number of useful elements in indexs will have decreased. The index of those pixels that couldn’t be labeled will remain inside indexs. If withrivers is non-zero, then pixels that are immediately touching more than one positive value will be given a GAL_LABEL_RIVER label.
Note that the `indexs->array` is not re-allocated to its new size at the end\(^{27}\). But since `indexs->dsize[0]` and `indexs->size` have new values after this function is returned, the extra elements just won’t be used until they are ultimately freed by `gal_data_free`.

Connectivity is a value between 1 (fewest number of neighbors) and the number of dimensions in the input (most number of neighbors). For example in a 2D dataset, a connectivity of 1 and 2 corresponds to 4-connected and 8-connected neighbors.

### 11.3.25 Convolution functions (convolve.h)

Convolution is a very common operation during data analysis and is thoroughly described as part of Gnuastro’s Section 6.3 [Convolve], page 306, program which is fully devoted to this job. Because of the complete introduction that was presented there, we will directly skip onto the currently available convolution functions in Gnuastro’s library.

As of this version, only spatial domain convolution is available in Gnuastro’s libraries. We haven’t had the time to liberate the frequency domain function convolution and deconvolution functions that are available in the Convolve program\(^{28}\).

```c
[Function]
gal_data_t *
gal_convolve_spatial (gal_data_t *tiles, gal_data_t *kernel, size_t numthreads, int edgecorrection, int convoverch)
```

Convolves the given `tiles` dataset (possibly a list of tiles, see Section 11.3.8.9 [List of `gal_data_t`], page 566, and Section 11.3.15 [Tessellation library (tile.h)], page 608) with `kernel` on `numthreads` threads. When `edgecorrection` is non-zero, it will correct for the edge dimming effects as discussed in Section 6.3.1.2 [Edges in the spatial domain], page 307.

- `tiles` can be a single/complete dataset, but in that case the speed will be very slow. Therefore, for larger images, it is recommended to give a list of tiles covering a dataset. To create a tessellation that fully covers an input image, you may use `gal_tile_full`, or `gal_tile_full_two_layers` to also define channels over your input dataset. These functions are discussed in Section 11.3.15.2 [Tile grid], page 614. You may then pass the list of tiles to this function. This is the recommended way to call this function because spatial domain convolution is slow and breaking the job into many small tiles and working on simultaneously on several threads can greatly speed up the processing.

- If the tiles are defined within a channel (a larger tile), by default convolution will be done within the channel, so pixels on the edge of a channel will not be affected by their neighbors that are in another channel. See Section 4.8 [Tessellation], page 189, for the necessity of channels in astronomical data analysis. This behavior may be disabled when `convoverch` is non-zero. In this case, it will ignore channel borders (if they exist) and mix all pixels that cover the kernel within the dataset.

---

\(^{27}\) Note that according to the GNU C Library, even a `realloc` to a smaller size can also cause a re-write of the whole array, which is not a cheap operation.

\(^{28}\) Hence any help would be greatly appreciated.
void [Function] 
gal_convolve_spatial_correct_ch_edge (gal_data_t *tiles, gal_data_t 
  *kernel, size_t numthreads, int edgecorrection, gal_data_t 
  *tocorrect) 

Correct the edges of channels in an already convolved image when it was initially 
convolved with gal_convolve_spatial and convoverch==0. In that case, strong 
boundaries might exist on the channel edges. So if you later need to remove those 
boundaries at later steps of your processing, you can call this function. It will only 
do convolution on the tiles that are near the edge and were effected by the channel 
borders. Other pixels in the image will not be touched. Hence, it is much faster.

11.3.26 Interpolation (interpolate.h) 

During data analysis, it happens that parts of the data cannot be given a value, but one 
is necessary for the higher-level analysis. For example a very bright star saturated part of 
your image and you need to fill in the saturated pixels with some values. Another common usage case are masked sky-lines in 1D spectra that similarly need to be assigned a value 
for higher-level analysis. In other situations, you might want a value in an arbitrary point: 
between the elements/pixels where you have data. The functions described in this section 
are for such operations.

The parametric interpolations discussed below are wrappers around the interpolation 
functions of the GNU Scientific Library (or GSL, see Section 3.1.1.1 [GNU Scientific 
Library], page 120). To identify the different GSL interpolation types, Gnuastro's 
gnuastro/interpolate.h header file contains macros that are discussed below. The GSL 
wrappers provided here are not yet complete because we are too busy. If you need them, 
please consider helping us in adding them to Gnuastro's library. Your contributions would 
be very welcome and appreciated.

GAL_INTERPOLATE_NEIGHBORS_METRIC_RADIAL [Macro] 
GAL_INTERPOLATE_NEIGHBORS_METRIC_MANHATTAN [Macro] 
GAL_INTERPOLATE_NEIGHBORS_METRIC_INVALID [Macro] 
The metric used to find distance for nearest neighbor interpolation. A radial metric 
uses the simple Euclidean function to find the distance between two pixels. A man-
hattan metric will always be an integer and is like steps (but is also much faster to 
calculate than radial metric because it doesn’t need a square root calculation).

GAL_INTERPOLATE_NEIGHBORS_FUNC_MIN [Macro] 
GAL_INTERPOLATE_NEIGHBORS_FUNC_MAX [Macro] 
GAL_INTERPOLATE_NEIGHBORS_FUNC_MEDIAN [Macro] 
GAL_INTERPOLATE_NEIGHBORS_FUNC_INVALID [Macro] 
The various types of nearest-neighbor interpolation functions for gal_interpolate_ 
neighbors. The names are descriptive for the operation they do, so we won’t go 
into much more detail here. The median operator will be one of the most used, but 
operators like the maximum are good to fill the center of saturated stars.
gal_data_t * gal_interpolate_neighbors (gal_data_t *input, struct gal_tile_two_layer_params *tl, uint8_t metric, size_t numneighbors, size_t numthreads, int onlyblank, int aslinkedlist, int function)
Interpolate the values in the input dataset using a calculated statistics from the distribution of their numneighbors closest neighbors. The desired statistics is determined from the func argument, which takes any of the GAL_INTERPOLATE_NEIGHBORS_FUNC Macros (see above). This function is non-parametric and thus agnostic to the input’s number of dimension or shape of the distribution.

Distance can be defined on different metrics that are identified through metric (taking values determined by the GAL_INTERPOLATE_NEIGHBORS_METRIC Macros described above). If onlyblank is non-zero, then only blank elements will be interpolated and pixels that already have a value will be left untouched. This function is multi-threaded and will run on numthreads threads (see gal_threads_number in Section 11.3.2 [Multithreaded programming (threads.h)], page 527).

tl is Gnuastro’s tessellation structure used to define tiles over an image and is fully described in Section 11.3.15.2 [Tile grid], page 614. When tl!=NULL, then it is assumed that the input->array contains one value per tile and interpolation will respect certain tessellation properties, for example to not interpolate over channel borders.

If several datasets have the same set of blank values, you don’t need to call this function multiple times. When aslinkedlist is non-zero, then input will be seen as a Section 11.3.8.9 [List of gal_data_t], page 566. In this case, the same neighbors will be used for all the datasets in the list. Of course, the values for each dataset will be different, so a different value will be written in each dataset, but the neighbor checking that is the most CPU intensive part will only be done once.

This is a non-parametric and robust function for interpolation. The interpolated values are also always within the range of the non-blank values and strong outliers do not get created. However, this type of interpolation must be used with care when there are gradients. This is because it is non-parametric and if there aren’t enough neighbors, step-like features can be created.

GAL_INTERPOLATE_1D_INVALID [Macro]
This is just a place-holder to manage errors.

GAL_INTERPOLATE_1D_LINEAR [Macro]
[From GSL:] Linear interpolation. This interpolation method does not require any additional memory.

GAL_INTERPOLATE_1D_POLYNOMIAL [Macro]
[From GSL:] Polynomial interpolation. This method should only be used for interpolating small numbers of points because polynomial interpolation introduces large oscillations, even for well-behaved datasets. The number of terms in the interpolating polynomial is equal to the number of points.

GAL_INTERPOLATE_1D_CSPLINE [Macro]
[From GSL:] Cubic spline with natural boundary conditions. The resulting curve is piecewise cubic on each interval, with matching first and second derivatives at the
supplied data-points. The second derivative is chosen to be zero at the first point and last point.

GAL_INTERPOLATE_1D_CSPLINE_PERIODIC  [Macro]
[From GSL:] Cubic spline with periodic boundary conditions. The resulting curve is piecewise cubic on each interval, with matching first and second derivatives at the supplied data-points. The derivatives at the first and last points are also matched. Note that the last point in the data must have the same y-value as the first point, otherwise the resulting periodic interpolation will have a discontinuity at the boundary.

GAL_INTERPOLATE_1D_AKIMA  [Macro]
[From GSL:] Non-rounded Akima spline with natural boundary conditions. This method uses the non-rounded corner algorithm of Wodicka.

GAL_INTERPOLATE_1D_AKIMA_PERIODIC  [Macro]
[From GSL:] Non-rounded Akima spline with periodic boundary conditions. This method uses the non-rounded corner algorithm of Wodicka.

GAL_INTERPOLATE_1D_STEFFEN  [Macro]
[From GSL:] Steffen’s method guarantees the monotonicity of the interpolating function between the given data points. Therefore, minima and maxima can only occur exactly at the data points, and there can never be spurious oscillations between data points. The interpolated function is piecewise cubic in each interval. The resulting curve and its first derivative are guaranteed to be continuous, but the second derivative may be discontinuous.

gsl_spline *  [Function]
gal_interpolate_1d_make_gsl_spline (gal.data_t *X, gal.data_t *Y, int type_1d)
Allocate and initialize a GNU Scientific Library (GSL) 1D gsl_spline structure using the non-blank elements of Y. type_1d identifies the interpolation scheme and must be one of the GAL_INTERPOLATE_1D_* macros defined above.

If X==NULL, the X-axis is assumed to be integers starting from zero (the index of each element in Y). Otherwise, the values in X will be used to initialize the interpolation structure. Note that when given, X must not contain any blank elements and it must be sorted (in increasing order).

Each interpolation scheme needs a minimum number of elements to successfully operate. If the number of non-blank values in Y is less than this number, this function will return a NULL pointer.

To be as generic and modular as possible, GSL’s tools are low-level. Therefore before doing the interpolation, many steps are necessary (like preparing your dataset, then allocating and initializing gsl_spline). The metadata available in Gnuastro’s Section 11.3.6.1 [Generic data container (gal.data_t)], page 542, make it easy to hide all those preparations within this function.

Once gsl_spline has been initialized by this function, the interpolation can be evaluated for any X value within the non-blank range of the input using gsl_spline_eval or gsl_spline_eval_e.

29 http://adsabs.harvard.edu/abs/1990A%26A...239..443S
For example in the small program below (sample-interp.c), we read the first two columns of the table in table.txt and feed them to this function to later estimate the values in the second column for three selected points. You can use Section 11.2 [BuildProgram], page 521, to compile and run this function, see Section 11.4 [Library demo programs], page 662, for more.

Contents of the table.txt file:

```
$ cat table.txt
0 0
1 2
3 6
4 8
6 12
8 16
9 18
```

Contents of the sample-interp.c file:

```
#include <stdio.h>
#include <stdlib.h>
#include <gnuastro/table.h>
#include <gnuastro/interpolate.h>

int main(void)
{
   size_t i;
   gal_data_t *X, *Y;
   gsl_spline *spline;
   gsl_interp_accel *acc;
   gal_list_str_t *cols=NULL;
   /* Change the values based on your input table. */
   double points[]={1.8, 2.5, 7};

   /* Read the first two columns from 'tab.txt'.
   IMPORTANT: the list is first-in-first-out, so the output
   column order is the inverse of the input order. */
   gal_list_str_add(&cols, "1", 0);
   gal_list_str_add(&cols, "2", 0);
   Y=gal_table_read("table.txt", NULL, NULL, cols,
                  GAL_TABLE_SEARCH_NAME, 0, 1, -1, 1, NULL);
   X=Y->next;

   /* Allocate the GSL interpolation accelerator and make the
   'gsl_spline' structure. */
   acc=gsl_interp_accel_alloc();
   spline=gal_interpolate_1d_make_gsl_spline(X, Y,
                  GAL_INTERPOLATE_1D_STEFFEN);
```
/* Calculate the respective value for all the given points, if 'spline' could be allocated. */
if(spline)
    for(i=0; i<(sizeof points)/(sizeof *points); ++i)
        printf("%f: %f\n", points[i],
            gsl_spline_eval(spline, points[i], acc));

/* Clean up and return. */
gal_data_free(X);
gal_data_free(Y);
gsl_spline_free(spline);
gsl_interp_accel_free(acc);
gal_list_str_free(cols, 0);
return EXIT_SUCCESS;
}

Compile and run this program with Section 11.2 [BuildProgram], page 521, to see the interpolation results for the three points within the program.

$ astbuildprog sample-interp.c --quiet
1.800000: 3.600000
2.500000: 5.000000
7.000000: 14.000000

void
gal_interpolate_1d_blank (gal_data_t *in, int type_1d)
    Fill the blank elements of in using the rest of the elements and the given interpolation.
The interpolation scheme can be set through type_1d, which accepts any of the GAL_INTERPOLATE_1D_* macros above. The interpolation is internally done in 64-bit floating point type (double). However the evaluated/interpolated values (originally blank) will be written (in in) with its original numeric datatype, using C’s standard type conversion.

By definition, interpolation is only defined “between” valid points. Therefore, if any number of elements on the start or end of the 1D array are blank, those elements will not be interpolated and will remain blank. To see if any blank (non-interpolated) elements remain, you can use gal_blank_present on in after this function is finished.

11.3.27 Git wrappers (git.h)

Git is one of the most common tools for version control and it can often be useful during development, for example see COMMIT keyword in Section 4.10 [Output FITS files], page 191. At installation time, Gnuastro will also check for the existence of libgit2, and store the value in the GAL_CONFIG_HAVE_LIBGIT2, see Section 11.3.1 [Configuration information (config.h)], page 526, and Section 3.1.2 [Optional dependencies], page 122. gnuastro/git.h includes gnuastro/config.h internally, so you won’t have to include both for this macro.
Chapter 11: Library 655

char * gal_git_describe ( )

When libgit2 is present and the program is called within a directory that is version controlled, this function will return a string containing the commit description (similar to Gnuastro’s unofficial version number, see Section 1.6 [Version numbering], page 7). If there are uncommitted changes in the running directory, it will add a ‘-dirty’ prefix to the description. When there is no tagged point in the previous commit, this function will return a uniquely abbreviated commit object as fallback. This function is used for generating the value of the COMMIT keyword in Section 4.10 [Output FITS files], page 191. The output string is similar to the output of the following command:

$ git describe --dirty --always

Space for the output string is allocated within this function, so after using the value you have to free the output string. If libgit2 is not installed or the program calling this function is not within a version controlled directory, then the output will be the NULL pointer.

11.3.28 Unit conversion library (units.h)

Datasets can contain values in various formats or units. The functions in this section are defined to facilitate the easy conversion between them and are declared in units.h. If there are certain conversions that are useful for your work, please get in touch.

int gal_units_extract_decimal (char *convert, const char *delimiter, double *args, size_t n)

Parse the input convert string with a certain delimiter (for example 01:23:45, where the delimiter is ":") as multiple numbers (for example 1,23,45) and write them as an array in the space that args is pointing to. The expected number of values in the string is specified by the n argument (3 in the example above).

If the function succeeds, it will return 1, otherwise it will return 0 and the values may not be fully written into args. If the number of values parsed in the string is different from n, this function will fail.

double gal_units_ra_to_degree (char *convert)

Convert the input Right Ascension (RA) string (in the format of hours, minutes and seconds either as _h_m_s or _:_:_:) to degrees (a single floating point number).

double gal_units_dec_to_degree (char *convert)

Convert the input Declination (Dec) string (in the format of degrees, arc-minutes and arc-seconds either as _d_m_s or _:_:_:) to degrees (a single floating point number).

char * gal_units_degree_to_ra (double decimal, int usecolon)

Convert the input Right Ascension (RA) degree (a single floating point number) to old/standard notation (in the format of hours, minutes and seconds of _h_m_s). If usecolon!=0, then the delimiters between the components will be colons: _:_:_.
char * gal_units_degree_to_dec (double decimal, int usecolon)
    Convert the input Declination (Dec) degree (a single floating point number) to old/standard notation (in the format of degrees, arc-minutes and arc-seconds of _d_m_s). If usecolon!=0, then the delimiters between the components will be colons: _:_:_.

double gal_units_counts_to_mag (double counts, double zeropoint)
    Convert counts to magnitudes through the given zero point. For more on the equation, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

double gal_units_mag_to_counts (double mag, double zeropoint)
    Convert magnitudes to counts through the given zero point. For more on the equation, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

double gal_units_counts_to_jy (double counts, double zeropoint_ab)
    Convert counts to Janskys through an AB magnitude-based zero point. For more on the equation, see Section 7.4.2 [Brightness, Flux, Magnitude and Surface brightness], page 394.

double gal_units_au_to_pc (double au)
    Convert the input value (assumed to be in Astronomical Units) to Parsecs. For the conversion equation, see the description of au-to-pc operator in Section 6.2.3 [Arithmetic operators], page 276.

double gal_units_pc_to_au (double pc)
    Convert the input value (assumed to be in Parsecs) to Astronomical Units (AUs). For the conversion equation, see the description of au-to-pc operator in Section 6.2.3 [Arithmetic operators], page 276.

double gal_units_ly_to_pc (double ly)
    Convert the input value (assumed to be in Light-years) to Parsecs. For the conversion equation, see the description of ly-to-pc operator in Section 6.2.3 [Arithmetic operators], page 276.

double gal_units_pc_to_ly (double pc)
    Convert the input value (assumed to be in Parsecs) to Light-years. For the conversion equation, see the description of ly-to-pc operator in Section 6.2.3 [Arithmetic operators], page 276.
double gal_units_ly_to_au (double ly)
    Convert the input value (assumed to be in Light-years) to Astronomical Units. For the conversion equation, see the description of \texttt{ly-to-pc} operator in Section 6.2.3 [Arithmetic operators], page 276.

double gal_units_au_to_ly (double au)
    Convert the input value (assumed to be in Astronomical Units) to Light-years. For the conversion equation, see the description of \texttt{ly-to-pc} operator in Section 6.2.3 [Arithmetic operators], page 276.

11.3.29 Spectral lines library (speclines.h)
Gnuastro’s library has the following macros and functions for dealing with spectral lines. All these functions are declared in \texttt{gnuastro/spectra.h}.

\begin{verbatim}
GAL_SPECLINES_INVALID [Macro]
GAL_SPECLINES_SIIRED [Macro]
GAL_SPECLINES_SII [Macro]
GAL_SPECLINES_SIIBLUE [Macro]
GAL_SPECLINES_NIIRED [Macro]
GAL_SPECLINES_NII [Macro]
GAL_SPECLINES_HALPHA [Macro]
GAL_SPECLINES_NIIBLUE [Macro]
GAL_SPECLINES_OIIIRED [Macro]
GAL_SPECLINES_OIII [Macro]
GAL_SPECLINES_OIIIBLUE [Macro]
GAL_SPECLINES_HBETA [Macro]
GAL_SPECLINES_HEIIRED [Macro]
GAL_SPECLINES_HEGAMMA [Macro]
GAL_SPECLINES_HDELTA [Macro]
GAL_SPECLINES_HEPSILON [Macro]
GAL_SPECLINES_NEIII [Macro]
GAL_SPECLINES_OIIRED [Macro]
GAL_SPECLINES_OII [Macro]
GAL_SPECLINES_OIIBLUE [Macro]
GAL_SPECLINES_BLIMIT [Macro]
GAL_SPECLINES_MGIIRE [Macro]
GAL_SPECLINES_MGII [Macro]
GAL_SPECLINES_MGIIBLUE [Macro]
GAL_SPECLINES_CIIRED [Macro]
GAL_SPECLINES_CI [Macro]
GAL_SPECLINES_CIIBLUE [Macro]
GAL_SPECLINES_HEIIBLUE [Macro]
GAL_SPECLINES_LYALPHA [Macro]
GAL_SPECLINES_LYLIMIT [Macro]
\end{verbatim}
Chapter 11: Library

GAL_SPECLINES_INVALID_MAX
Internal values/identifiers for specific spectral lines as is clear from their names. Note the first and last one, they can be used when parsing the lines automatically: both don’t correspond to any line, but their integer values correspond to the two integers just before and after the first and last line identifier.

GAL_SPECLINES_INVALID has a value of zero, and allows you to have a fixed integer which never corresponds to a line. GAL_SPECLINES_INVALID_MAX is the total number of pre-defined lines, plus one. So you can parse all the known lines with a for loop like this:

```c
for(i=1;i<GAL_SPECLINES_INVALID_MAX;++i)
```

Wavelength (in Angstroms) of the named lines.

GAL_SPECLINES_NAME_SIIRED [Macro]
GAL_SPECLINES_NAME_SII [Macro]
GAL_SPECLINES_NAME_SIIBLUE [Macro]
GAL_SPECLINES_NAME_NIIRED [Macro]
GAL_SPECLINES_NAME_NII [Macro]
GAL_SPECLINES_NAME_HALPHA [Macro]
GAL_SPECLINES_NAME_NIIBLUE [Macro]
GAL_SPECLINES_NAME_OIIIRED [Macro]
GAL_SPECLINES_NAME_OIII [Macro]
GAL_SPECLINES_NAME_OIIIBLUE [Macro]
GAL_SPECLINES_NAME_HBETA [Macro]
GAL_SPECLINES_NAME_HEIIRED [Macro]
GAL_SPECLINES_NAME_HGAMMA [Macro]
GAL_SPECLINES_NAME_HDELTA [Macro]
GAL_SPECLINES_NAME_HEPSILON [Macro]
GAL_SPECLINES_NAME_NEIII [Macro]
GAL_SPECLINES_NAME_OIIIRED [Macro]
GAL_SPECLINES_NAME_OII [Macro]
GAL_SPECLINES_NAME_OIIBLUE [Macro]
GAL_SPECLINES_NAME_BLIMIT [Macro]
GAL_SPECLINES_NAME_MGIIRED [Macro]
GAL_SPECLINES_NAME_MGIIBLUE [Macro]
GAL_SPECLINES_NAME_CIIIRED [Macro]
GAL_SPECLINES_NAME_CIII [Macro]
GAL_SPECLINES_NAME_CIIIBLUE [Macro]
GAL_SPECLINES_NAME_HEIIBLUE [Macro]
GAL_SPECLINES_NAME_LYALPHA [Macro]
GAL_SPECLINES_NAME_LYLIMIT [Macro]
Names (as literal strings without any space, all in small-caps) that can be used to refer to the lines in your program and converted to and from line identifiers using the functions below.

char *
gal_speclines_line_name (int linecode)
    Return the literal string of the given spectral line identifier Macro (for example GAL_SPECLINES_NAME_HALPHA or GAL_SPECLINES_NAME_LYLIMIT).

int

gal_speclines_line_code (char *name)
    Return the spectral line identifier of the given standard name (for example GAL_SPECLINES_NAME_HALPHA or GAL_SPECLINES_NAME_LYLIMIT).

double

gal_speclines_line_angstrom (int linecode)
    Return the wavelength (in Angstroms) of the given line.

double

gal_speclines_line_redshift (double obsline, double restline)
    Return the redshift where the observed wavelength (obsline) was emitted from (if its restframe wavelength was restline).
double gal_speclines_line_redshift_code (double obsline, int linecode)
    
    Return the redshift where the observed wavelength (obsline) was emitted from (assuming its a specific spectra line, identified with linecode).

11.3.30 Cosmology library (cosmology.h)

This library does the main cosmological calculations that are commonly necessary in extra-galactic astronomical studies. The main variable in this context is the redshift (z). The cosmological input parameters in the functions below are H0, o_lambda_0, o_matter_0, o_radiation_0 which respectively represent the current (at redshift 0) expansion rate (Hubble constant in units of km/sec/Mpc), cosmological constant (Λ), matter and radiation densities.

All these functions are declared in gnuastro/cosmology.h. For a more extended introduction/discussion of the cosmological parameters, please see Section 9.1 [CosmicCalculator], page 469.

double gal_cosmology_age (double z, double H0, double o_lambda_0, double o_matter_0, double o_radiation_0)
    
    Returns the age of the universe at redshift z in units of Giga years.

double gal_cosmology_proper_distance (double z, double H0, double o_lambda_0, double o_matter_0, double o_radiation_0)
    
    Returns the proper distance to an object at redshift z in units of Mega parsecs.

double gal_cosmology_comoving_volume (double z, double H0, double o_lambda_0, double o_matter_0, double o_radiation_0)
    
    Returns the comoving volume over 4pi stradian to z in units of Mega parsecs cube.

double gal_cosmology_critical_density (double z, double H0, double o_lambda_0, double o_matter_0, double o_radiation_0)
    
    Returns the critical density at redshift z in units of g/cm$^3$.

double gal_cosmology_angular_distance (double z, double H0, double o_lambda_0, double o_matter_0, double o_radiation_0)
    
    Return the angular diameter distance to an object at redshift z in units of Mega parsecs.

double gal_cosmology_luminosity_distance (double z, double H0, double o_lambda_0, double o_matter_0, double o_radiation_0)
    
    Return the luminosity diameter distance to an object at redshift z in units of Mega parsecs.
double [Function] 
gal_cosmology_distance_modulus (double z, double H0, double o_lambda_0, double o_matter_0, double o_radiation_0) 
Return the distance modulus at redshift z (with no units).

double [Function] 
gal_cosmology_to_absolute_mag (double z, double H0, double o_lambda_0, double o_matter_0, double o_radiation_0) 
Return the conversion from apparent to absolute magnitude for an object at redshift z. This value has to be added to the apparent magnitude to give the absolute magnitude of an object at redshift z.

double [Function] 
gal_cosmology_velocity_from_z (double z) 
Return the velocity (in km/s) corresponding to the given redshift (z).

double [Function] 
gal_cosmology_z_from_velocity (double v) 
Return the redshift corresponding to the given velocity (v in km/s).

11.3.31 SAO DS9 library (ds9.h)
This library operates on the output files of SAO DS9. SAO DS9 is one of the most commonly used FITS image and cube viewers today with an easy to use graphic user interface (GUI), see Section B.1 [SAO DS9], page 702. But besides merely opening FITS data, it can also produce certain kinds of files that can be useful in common analysis. For example, on DS9’s GUI, it is very easy to define a (possibly complex) polygon as a “region”. You can then save that “region” into a file and using the functions below, feed the polygon into Gnuastro’s programs (or your custom programs).

GAL_DS9_COORD_MODE_IMG [Macro]
GAL_DS9_COORD_MODE_WCS [Macro]
GAL_DS9_COORD_MODE_INVALID [Macro]
Macros to identify the coordinate mode of the DS9 file. Their names are sufficiently descriptive. The last one (INVALID) is for sanity checks (for example, to know if the mode is already selected).

gal_data_t * [Function] 
gal_ds9_reg_read_polygon (char *filename) 
Returns an allocated generic data container (gal_data_t, with an array of GAL_TYPE_FLOAT64) containing the vertices of a polygon within the SAO DS9 region file given by *filename. Since SAO DS9 region files are 2 dimensional, if there are N vertices in the SAO DS9 region file, the returned dataset will have 2 × N elements (first two elements belonging to first vertex, and etc).

The mode to interpret the vertex coordinates is also read from the SAO DS9 region file and written into the status attribute of the output gal_data_t. The coordinate mode can be one of the GAL_DS9_COORD_MODE_* macros, mentioned above.

30 https://sites.google.com/cfa.harvard.edu/saoimageds9
It is assumed that the file begins with # Region file format: DS9 and it has two more lines (at least): a line containing the mode of the coordinates (the line should only contain either fk5 or image), a line with the polygon vertices following this format: polygon(V1X,V1Y,V2X,V2Y,...) where V1X and V1Y are the horizontal and vertical coordinates of the first vertex, and so on.

For example, here is a minimal acceptable SAO DS9 region file:

```plaintext
# Region file format: DS9
fk5
polygon(53.187414,-27.779152,53.159507,-27.759633,...)
```

11.4 Library demo programs

In this final section of Chapter 11 [Library], page 513, we give some example Gnuastro programs to demonstrate various features in the library. All these programs have been tested and once Gnuastro is installed you can compile and run them with Gnuastro's Section 11.2 [BuildProgram], page 521, program that will take care of linking issues. If you don't have any FITS file to experiment on, you can use those that are generated by Gnuastro after make check in the tests/ directory, see Section 1.1 [Quick start], page 1.

11.4.1 Library demo - reading a FITS image

The following simple program demonstrates how to read a FITS image into memory and use the void *array pointer in of Section 11.3.6.1 [Generic data container (gal_data_t)], page 542. For easy linking/compilation of this program along with a first run see Section 11.2 [BuildProgram], page 521. Before running, also change the filename and hdu variable values to specify an existing FITS file and/or extension/HDU.

This is just intended to demonstrate how to use the array pointer of gal_data_t. Hence it doesn’t do important sanity checks, for example in real datasets you may also have blank pixels. In such cases, this program will return a NaN value (see Section 6.1.3 [Blank pixels], page 262). So for general statistical information of a dataset, it is much better to use Gnuastro's Section 7.1 [Statistics], page 337, program which can deal with blank pixels and many other issues in a generic dataset.

```c
#include <stdio.h>
#include <stdlib.h>
#include <gnuastro/fits.h> /* includes gnuastro's data.h and type.h */
#include <gnuastro/statistics.h>

int
main(void)
{
    size_t i;
    float *farray;
    double sum=0.0f;
    gal_data_t *image;
    char *filename="img.fits", *hdu="1";
```
Chapter 11: Library       663

/* Read 'img.fits' (HDU: 1) as a float32 array. */
image=gal_fits_img_read_to_type(filename, hdu, GAL_TYPE_FLOAT32, -1, 1);

/* Use the allocated space as a single precision floating point array (recall that 'image->array' has 'void *'
type, so it is not directly usable. */
farray=image->array;

/* Calculate the sum of all the values. */
for(i=0; i<image->size; ++i)
    sum += farray[i];

/* Report the sum. */
printf("Sum of values in %s (hdu %s) is: %f\n", filename, hdu, sum);

/* Clean up and return. */
gal_data_free(image);
return EXIT_SUCCESS;
}

11.4.2 Library demo - inspecting neighbors

The following simple program shows how you can inspect the neighbors of a pixel using the
GAL_DIMENSION_NEIGHBOR_OP function-like macro that was introduced in Section 11.3.7
[Dimensions (dimension.h)], page 550. For easy linking/compilation of this program along
with a first run see Section 11.2 [BuildProgram], page 521. Before running, also change
the file name and HDU (first and second arguments to gal_fits_img_read_to_type) to
specify an existing FITS file and/or extension/HDU.

#include <stdio.h>
#include <stdlib.h>
#include <gnuastro/fits.h>
#include <gnuastro/dimension.h>

int
main(void)
{
    double sum;
    float *array;
    size_t i, num, *dinc;
    gal_data_t *input=gal_fits_img_read_to_type("input.fits", "1",
        GAL_TYPE_FLOAT32, -1, 1);
/* To avoid the 'void *' pointer and have 'dinc'. */
array=input->array;
dinc=gal_dimension_increment(input->ndim, input->dsize);

/* Go over all the pixels. */
for(i=0;i<input->size;++i)
{
    num=0;
    sum=0.0f;
    GAL_DIMENSION_NEIGHBOR_OP( i, input->ndim, input->dsize,
                               input->ndim, dinc,
                               {++num; sum+=array[nind];}
    );
    printf("%zu: num: %zu, sum: %f\n", i, num, sum);
}

/* Clean up and return. */
gal_data_free(input);
return EXIT_SUCCESS;

11.4.3 Library demo - multi-threaded operation

The following simple program shows how to use Gnuastro to simplify spinning off threads
and distributing different jobs between the threads. The relevant thread-related functions
are defined in Section 11.3.2.2 [Gnuastro's thread related functions], page 529. For easy
linking/compilation of this program, along with a first run, see Gnuastro's Section 11.2
[BuildProgram], page 521. Before running, also change the filename and hdu variable
values to specify an existing FITS file and/or extension/HDU.

This is a very simple program to open a FITS image, distribute its pixels between
different threads and print the value of each pixel and the thread it was assigned to. The
actual operation is very simple (and would not usually be done with threads in a real-life
program). It is intentionally chosen to put more focus on the important steps in spinning
of threads and how the worker function (which is called by each thread) can identify the
job-IDs it should work on.

For example, instead of an array of pixels, you can define an array of tiles or any other
case-specific structures as separate targets. The important thing is that each action
should have its own unique ID (counting from zero, as is done in an array in C). You can
then follow the process below and use each thread to work on all the targets that are assigned
to it. Recall that spinning-off threads is its self an expensive process and we don’t want to
spin-off one thread for each target (see the description of gal_threads_dist_in_threads
in Section 11.3.2.2 [Gnuastro's thread related functions], page 529.

There are many (more complicated, real-world) examples of using gal_threads_spin_off
in Gnuastro’s actual source code, you can see them by searching for the gal_threads_spin_off
function from the top source (after unpacking the tarball) directory (for example
with this command):

    $ grep -r gal_threads_spin_off ./

The code of this demonstration program is shown below. This program was also built and run when you ran `make check` during the building of Gnuastro (tests/lib/multithread.c), so it is already tested for your system and you can safely use it as a guide.

```c
#include <stdio.h>
#include <stdlib.h>

#include <gnuastro/fits.h>
#include <gnuastro/threads.h>

/* This structure can keep all information you want to pass onto the
 * worker function on each thread. */
struct params
{
    gal_data_t *image; /* Dataset to print values of. */
};

/* This is the main worker function which will be called by the
 * different threads. 'gal_threads_params' is defined in
 * 'gnuastro/threads.h' and contains the pointer to the parameter we
 * want. Note that the input argument and returned value of this
 * function always must have 'void *' type. */
void *
worker_on_thread(void *in_prm)
{
    /* Low-level definitions to be done first. */
    struct gal_threads_params *tprm=(struct gal_threads_params *)in_prm;
    struct params *p=(struct params *)tprm->params;

    /* Subsequent definitions. */
    float *array=p->image->array;
    size_t i, index, *dsize=p->image->dsize;

    /* Go over all the actions (pixels in this case) that were assigned
     * to this thread. */
    for(i=0; tprm->indexes[i] != GAL_BLANK_SIZE_T; ++i)
    {
        /* For easy reading. */
        index = tprm->indexes[i];
    }
```
Chapter 11: Library

```
/* Print the information. */
printf("(%zu, %zu) on thread %zu: %g\n",
       index%dsize[1]+1,
       index/dsize[1]+1, tprm->id, array[index]);
}

/* Wait for all the other threads to finish, then return. */
if(tprm->b) pthread_barrier_wait(tprm->b);
return NULL;
}

/* High-level function (called by the operating system). */
int
main(void)
{
  struct params p;
  char *filename="input.fits", *hdu="1";
  size_t numthreads=gal_threads_number();

  /* We are using * '-1' for 'minmapsize' to ensure that the image is
   * read into * memory and '1' for 'quietmmap' (which can also be
   * zero), see the "Memory management" section in the book. */
  int quietmmap=1;
  size_t minmapsize=-1;

  /* Read the image into memory as a float32 data type. */
  p.image=gal_fits_img_read_to_type(filename, hdu, GAL_TYPE_FLOAT32,
                                  minmapsize, quietmmap);

  /* Print some basic information before the actual contents: */
  printf("Pixel values of %s (HDU: %s) on %zu threads.\n", filename,
         hdu, numthreads);
  printf("Used to check the compiled library's capability in opening "
         "a FITS file, and also spinning-off threads.\n\n");;

  /* A small sanity check: this is only intended for 2D arrays (to
   * print the coordinates of each pixel). */
  if(p.image->ndim!=2)
  {
    fprintf(stderr, "only 2D images are supported.");
  }
```
Chapter 11: Library

exit(EXIT_FAILURE);
}

/* Spin-off the threads and do the processing on each thread. */
gal_threads_spin_off(worker_on_thread, &p, p.image->size, numthreads,
                     minmapsize, quietmmap);

/* Clean up and return. */
gal_data_free(p.image);
return EXIT_SUCCESS;
}

11.4.4 Library demo - reading and writing table columns

Tables are some of the most common inputs to, and outputs of programs. This section contains a small program for reading and writing tables using the constructs described in Section 11.3.10 [Table input output (table.h)], page 568. For easy linking/compilation of this program, along with a first run, see Gnuastro’s Section 11.2 [BuildProgram], page 521. Before running, also set the following file and column names in the first two lines of main. The input and output names may be .txt and .fits tables, gal_table_read and gal_table_write will be able to write to both formats. For plain text tables see Section 4.7.2 [Gnuastro text table format], page 185.

This example program reads three columns from a table. The first two columns are selected by their name (NAME1 and NAME2) and the third is selected by its number: column 10 (counting from 1). Gnuastro’s column selection is discussed in Section 4.7.3 [Selecting table columns], page 188. The first and second columns can be any type, but this program will convert them to int32_t and float for its internal usage respectively. However, the third column must be double for this program. So if it isn’t, the program will abort with an error. Having the columns in memory, it will print them out along with their sum (just a simple application, you can do whatever you want at this stage). Reading the table finishes here.

The rest of the program is a demonstration of writing a table. While parsing the rows, this program will change the first column (to be counters) and multiply the second by 10 (so the output will be different). Then it will define the order of the output columns by setting the next element (to create a Section 11.3.8.9 [List of gal_data_t], page 566). Before writing, this function will also set names for the columns (units and comments can be defined in a similar manner). Writing the columns to a file is then done through a simple call to gal_table_write.

The operations that are shown in this example program are not necessary all the time. For example, in many cases, you know the numerical data type of the column before writing your program (see Section 4.5 [Numeric data types], page 178), so type checking and copying to a specific type won’t be necessary.

#include <stdio.h>
#include <stdlib.h>
#include <gnuastro/table.h>

int
main(void)
{
    /* File names and column names (which may also be numbers). */
    char *c1_name="NAME1", *c2_name="NAME2", *c3_name="10";
    char *inname="input.fits", *hdu="1", *outname="out.fits";

    /* Internal parameters. */
    float *array2;
    double *array3;
    int32_t *array1;
    size_t i, counter=0;
    gal_data_t *c1, *c2;
    gal_data_t tmp, *col, *columns;
    gal_list_str_t *column_ids=NULL;

    /* Define the columns to read. */
    gal_list_str_add(&column_ids, c1_name, 0);
    gal_list_str_add(&column_ids, c2_name, 0);
    gal_list_str_add(&column_ids, c3_name, 0);

    /* The columns were added in reverse, so correct it. */
    gal_list_str_reverse(&column_ids);

    /* Read the desired columns. */
    columns = gal_table_read(inname, hdu, column_ids,
                            GAL_TABLE_SEARCH_NAME, 1, -1, 1, NULL);

    /* Go over the columns, we'll assume that you don't know their type
    * a-priori, so we'll check */
    counter=1;
    for(col=columns; col!=NULL; col=col->next)
        switch(counter++)
        {
        case 1: /* First column: we want it as int32_t. */
            c1=gal_data_copy_to_new_type(col, GAL_TYPE_INT32);
            array1 = c1->array;
            break;

        case 2: /* Second column: we want it as float. */
            c2=gal_data_copy_to_new_type(col, GAL_TYPE_FLOAT32);
            array2 = c2->array;
            break;

        case 3: /* Third column: it MUST be double. */
if(col->type!=GAL_TYPE_FLOAT64)
{
    fprintf(stderr, "Column %s must be float64 type, it is "
            "%s", c3_name, gal_type_name(col->type, 1));
    exit(EXIT_FAILURE);
}
array3 = col->array;
break;
}

/* As an example application we'll just print them out. In the
 * meantime (just for a simple demonstration), change the first
 * array value to the counter and multiply the second by 10. */
for(i=0;i<c1->size;++i)
{
    printf("%zu: %d + %f + %f = %f\n", i+1, array1[i], array2[i],
            array3[i], array1[i]+array2[i]+array3[i]);
    array1[i] = i+1;
    array2[i] *= 10;
}

/* Link the first two columns as a list. */
c1->next = c2;
c2->next = NULL;

/* Set names for the columns and write them out. */
c1->name = "COUNTER";
c2->name = "VALUE";
gal_table_write(c1, NULL, NULL, GAL_TABLE_FORMAT_BFITS, outname,
                "MY-COLUMNS", 0);

/* The names weren't allocated, so to avoid cleaning-up problems,
 * we'll set them to NULL. */
c1->name = c2->name = NULL;

/* Clean up and return. */
gal_data_free(c1);
gal_data_free(c2);
gal_list_data_free(columns);
gal_list_str_free(column_ids, 0); /* strings weren't allocated. */
return EXIT_SUCCESS;"
12 Developing

The basic idea of GNU Astronomy Utilities is for an interested astronomer to be able to easily understand the code of any of the programs or libraries, be able to modify the code if s/he feels there is an improvement and finally, to be able to add new programs or libraries for their own benefit, and the larger community if they are willing to share it. In short, we hope that at least from the software point of view, the “obscurantist faith in the expert’s special skill and in his personal knowledge and authority” can be broken, see Section 1.2 [Gnuastro manifesto: Science and its tools], page 2. With this aim in mind, Gnuastro was designed to have a very basic, simple, and easy to understand architecture for any interested inquirer.

This chapter starts with very general design choices, in particular Section 12.1 [Why C programming language?], page 670, and Section 12.2 [Program design philosophy], page 672. It will then get a little more technical about the Gnuastro code and file/directory structure in Section 12.3 [Coding conventions], page 673, and Section 12.4 [Program source], page 677. Section 12.4.2 [The TEMPLATE program], page 680, discusses a minimal (and working) template to help in creating new programs or easier learning of a program’s internal structure. Some other general issues about documentation, building and debugging are then discussed. This chapter concludes with how you can learn about the development and get involved in Section 12.10 [Gnuastro project webpage], page 691, Section 12.11 [Developing mailing lists], page 692, and Section 12.12 [Contributing to Gnuastro], page 693.

12.1 Why C programming language?

Currently the programming languages that are commonly used in scientific applications are C++\(^1\), Java\(^2\), Python\(^3\), and Julia\(^4\) (which is a newcomer but swiftly gaining ground). One of the main reasons behind choosing these is their high-level abstractions. However, GNU Astronomy Utilities is fully written in the C programming language\(^5\). The reasons can be summarized with simplicity, portability and efficiency/speed. All four are very important in a scientific software and we will discuss them below.

Simplicity can best be demonstrated in a comparison of the main books of C++ and C. The “C programming language”\(^6\) book, written by the authors of C, is only 286 pages and covers a very good fraction of the language, it has also remained unchanged from 1988. C is the main programming language of nearly all operating systems and there is no plan of any significant update. On the other hand, the most recent “C++ programming language”\(^7\) book, also written by its author, has 1366 pages and its fourth edition came out in 2013! As discussed in Section 1.2 [Gnuastro manifesto: Science and its tools], page 2, it is very important for other scientists to be able to readily read the code of a program at their will with minimum requirements.

---

1. https://isocpp.org/
3. https://www.python.org/
4. https://julialang.org/
In C++ or Java, inheritance in the object oriented programming paradigm and their internal functions make the code very easy to write for a programmer who is deeply invested in those objects and understands all their relations well. But it simultaneously makes reading the program for a first time reader (a curious scientist who wants to know only how a small step was done) extremely hard. Before understanding the methods, the scientist has to invest a lot of time and energy in understanding those objects and their relations. But in C, everything is done with basic language types for example \texttt{ints} or \texttt{floats} and their pointers to define arrays. So when an outside reader is only interested in one part of the program, that part is all they have to understand.

Recently it is also becoming common to write scientific software in Python, or a combination of it with C or C++. Python is a high level scripting language which doesn’t need compilation. It is very useful when you want to do something on the go and don’t want to be halted by the troubles of compiling, linking, memory checking, etc. When the datasets are small and the job is temporary, this ability of Python is great and is highly encouraged. A very good example might be plotting, in which Python is undoubtedly one of the best.

But as the data sets increase in size and the processing becomes more complicated, the speed of Python scripts significantly decrease. So when the program doesn't change too often and is widely used in a large community, mostly on large data sets (like astronomical images), using Python will waste a lot of valuable research-hours. It is possible to wrap C or C++ functions with Python to fix the speed issue. But this creates further complexity, because the interested scientist has to master two programming languages and their connection (which is not trivial).

Like C++, Python is object oriented, so as explained above, it needs a high level of experience with that particular program to reasonably understand its inner workings. To make things worse, since it is mainly for on-the-go programming\footnote{Note that Python is good for fast programming, not fast programs.}, it can undergo significant changes. One recent example is how Python 2.x and Python 3.x are not compatible. Lots of research teams that invested heavily in Python 2.x cannot benefit from Python 3.x or future versions any more. Some converters are available, but since they are automatic, lots of complications might arise in the conversion\footnote{For example see Jenness (2017) (https://arxiv.org/abs/1712.00461) which describes how LSST is managing the transition.}. If a research project begins using Python 3.x today, there is no telling how compatible their investments will be when Python 4.x or 5.x will come out.

Java is also fully object-oriented, but uses a different paradigm: its compilation generates a hardware-independent \textit{bytecode}, and a \textit{Java Virtual Machine (JVM)} is required for the actual execution of this bytecode on a computer. Java also evolved with time, and tried to remain backward compatible, but inevitably some evolutions required discontinuities and replacements of a few Java components which were first declared as becoming \textit{deprecated}, and removed from later versions.

This stems from the core principles of high-level languages like Python or Java: that they evolve significantly on the scale of roughly 5 to 10 years. They are therefore useful when you want to solve a short-term problem and you are ready to pay the high cost of keeping your software up to date with all the changes in the language. This is fine for private companies, but usually too expensive for scientific projects that have limited funding for a
fixed period. As a result, the reproducibility of the result (ability to regenerate the result in the future, which is a core principal of any scientific result) and re-usability of all the investments that went into the science software will be lost to future generations! Rebuilding all the dependencies of a software in an obsolete language is not easy, or even not possible. Future-proof code (as long as current operating systems will be used) is therefore written in C.

The portability of C is best demonstrated by the fact that C++, Java and Python are part of the C-family of programming languages which also include Julia, Perl, and many other languages. C libraries can be immediately included in C++, and it is easy to write wrappers for them in all C-family programming languages. This will allow other scientists to benefit from C libraries using any C-family language that they prefer. As a result, Gnuastro’s library is already usable in C and C++, and wrappers will be\(^\text{10}\) added for higher-level languages like Python, Julia and Java.

The final reason was speed. This is another very important aspect of C which is not independent of simplicity (first reason discussed above). The abstractions provided by the higher-level languages (which also makes learning them harder for a newcomer) come at the cost of speed. Since C is a low-level language\(^\text{11}\) (closer to the hardware), it has a direct access to the CPU\(^\text{12}\), is generally considered as being faster in its execution, and is much less complex for both the human reader and the computer. The benefits of simplicity for a human were discussed above. Simplicity for the computer translates into more efficient (faster) programs. This creates a much closer relation between the scientist/programmer (or their program) and the actual data and processing. The GNU coding standards\(^\text{13}\) also encourage the use of C over all other languages when generality of usage and “high speed” is desired.

### 12.2 Program design philosophy

The core processing functions of each program (and all libraries) are written mostly with the basic ISO C90 standard. We do make lots of use of the GNU additions to the C language in the GNU C library\(^\text{14}\), but these functions are mainly used in the user interface functions (reading your inputs and preparing them prior to or after the analysis). The actual algorithms, which most scientists would be more interested in, are much more closer to ISO C90. For this reason, program source files that deal with user interface issues and those doing the actual processing are clearly separated, see Section 12.4 [Program source], page 677. If anything particular to the GNU C library is used in the processing functions, it is explained in the comments in between the code.

All the Gnuastro programs provide very low level and modular operations (modeled on GNU Coreutils). Almost all the basic command-line programs like \texttt{ls}, \texttt{cp} or \texttt{rm} on

---

\(^{10}\) \url{http://savannah.gnu.org/task/?13786}  
\(^{11}\) Low-level languages are those that directly operate the hardware like assembly languages. So C is actually a high-level language, but it can be considered one of the lowest-level languages among all high-level languages.  
\(^{12}\) for instance the \texttt{long double} numbers with at least 64-bit mantissa are not accessible in Python or Java.  
\(^{13}\) \url{http://www.gnu.org/prep/standards/}  
\(^{14}\) Gnuastro uses many GNU additions to the C library. However, thanks to the GNU Portability library (Gnulib) which is included in the Gnuastro tarball, users of non-GNU/Linux operating systems can also benefit from all these features when using Gnuastro.
GNU/Linux operating systems are part of GNU Coreutils. This enables you to use shell scripting languages (for example GNU Bash) to operate on a large number of files or do very complex things through the creative combinations of these tools that the authors had never dreamed of. We have put a few simple examples in Chapter 2 [Tutorials], page 17.

For example all the analysis output can be saved as ASCII tables which can be fed into your favorite plotting program to inspect visually. Python’s Matplotlib is very useful for fast plotting of the tables to immediately check your results. If you want to include the plots in a document, you can use the PGFplots package within \LaTeX, no attempt is made to include such operations in Gnuastro. In short, Bash can act as a glue to connect the inputs and outputs of all these various Gnuastro programs (and other programs) in any fashion. Of course, Gnuastro’s programs are just front-ends to the main workhorse (Section 11.3 [Gnuastro library], page 525), allowing a user to create their own programs (for example with Section 11.2 [BuildProgram], page 521). So once the functions within programs become mature enough, they will be moved within the libraries for even more general applications.

The advantage of this architecture is that the programs become small and transparent: the starting and finishing point of every program is clearly demarcated. For nearly all operations on a modern computer (fast file input-output) with a modest level of complexity, the read/write speed is insignificant compared to the actual processing a program does. Therefore the complexity which arises from sharing memory in a large application is simply not worth the speed gain. Gnuastro’s design is heavily influenced from Eric Raymond’s “The Art of Unix Programming”\textsuperscript{15} which beautifully describes the design philosophy and practice which lead to the success of Unix-based operating systems\textsuperscript{16}.

\section*{12.3 Coding conventions}

In Gnuastro, we try our best to follow the GNU coding standards. Added to those, Gnuastro defines the following conventions. It is very important for readability that the whole package follows the same convention.

- The code must be easy to read by eye. So when the order of several lines within a function does not matter (for example when defining variables at the start of a function). You should put the lines in the order of increasing length and group the variables with similar types such that this half-pyramid of declarations becomes most visible. If the reader is interested, a simple search will show them the variable they are interested in. However, this visual aid greatly helps in general inspections of the code and help the reader get a grip of the function’s processing.

- A function that cannot be fully displayed (vertically) in your monitor is probably too long and may be more useful if it is broken up into multiple functions. 40 lines is usually a good reference. When the start and end of a function are clearly visible in one glance, the function is much more easier to understand. This is most important for low-level functions (which usually define a lot of variables). Low-level functions do most of the processing, they will also be the most interesting part of a program for an inquiring astronomer. This convention is less important for higher level functions.


\textsuperscript{16} KISS principle: Keep It Simple, Stupid!
that don’t define too many variables and whose only purpose is to run the lower-level functions in a specific order and with checks.

In general you can be very liberal in breaking up the functions into smaller parts, the GNU Compiler Collection (GCC) will automatically compile the functions as inline functions when the optimizations are turned on. So you don’t have to worry about decreasing the speed. By default Gnuastro will compile with the `-O3` optimization flag.

- All Gnuastro hand-written text files (C source code, Texinfo documentation source, and version control commit messages) should not exceed 75 characters per line. Monitors today are certainly much wider, but with this limit, reading the functions becomes much more easier. Also for the developers, it allows multiple files (or multiple views of one file) to be displayed beside each other on wide monitors.

Emacs’s buffers are excellent for this capability, setting a buffer width of 80 with `C-u 80 C-x 3` will allow you to view and work on several files or different parts of one file using the wide monitors common today. Emacs buffers can also be used as a shell prompt and compile the program (with `M-x compile`), and 80 characters is the default width in most terminal emulators. If you use Emacs, Gnuastro sets the 75 character `fill-column` variable automatically for you, see cartouche below.

For long comments you can use press `Alt-q` in Emacs to separate them into separate lines automatically. For long literal strings, you can use the fact that in C, two strings immediately after each other are concatenated, for example "The first part, " "and the second part.". Note the space character in the end of the first part. Since they are now separated, you can easily break a long literal string into several lines and adhere to the maximum 75 character line length policy.

- The headers required by each source file (ending with `.c`) should be defined inside of it. All the headers a complete program needs should not be stacked in another header to include in all source files (for example `main.h`). Although most ‘professional’ programmers choose this single header method, Gnuastro is primarily written for professional/inquisitive astronomers (who are generally amateur programmers). The list of header files included provides valuable general information and helps the reader.

  `main.h` may only include the header file(s) that define types that the main program structure needs, see `main.h` in Section 12.4 [Program source], page 677. Those particular header files that are included in `main.h` can of course be ignored (not included) in separate source files.

- The headers should be classified (by an empty line) into separate groups:

  1. `#include <config.h>`: This must be the first code line (not commented or blank) in each source file within Gnuastro. It sets macros that the GNU Portability Library (Gnulib) will use for a unified environment (GNU C Library), even when the user is building on a system that doesn’t use the GNU C library.

  2. The C library header files, for example `stdio.h`, `stdlib.h`, or `math.h`.

  3. Installed library header files, including Gnuastro’s installed headers (for example `cfitsio.h` or `gsl/gsl_rng.h`, or `gnuastro/fits.h`).

  4. Gnuastro’s internal headers (that are not installed), for example `gnuastro-internal/options.h`.

  5. For programs, the `main.h` file (which is needed by the next group of headers).
6. That particular program’s header files, for example `mkprof.h`, or `noisechisel.h`. As much as order does not matter when you include the header of each group, sort them by length, as described above.

- All function names, variables, etc should be in lower case. Macros and constant global `enums` should be in upper case.
- For the naming of exported header files, functions, variables, macros, and library functions, we adopt similar conventions to those used by the GNU Scientific Library (GSL)\(^\text{17}\). In particular, in order to avoid clashes with the names of functions and variables coming from other libraries the name-space ‘`gal_`’ is prefixed to them. GAL stands for GNU Astronomy Library.
- All installed header files should be in the `lib/gnuastro` directory (under the top Gnuastro source directory). After installation, they will be put in the `$prefix/include/gnuastro` directory (see Section 3.3.1.2 [Installation directory], page 139, for `$prefix`). Therefore with this convention Gnuastro’s headers can be included in internal (to Gnuastro) and external (a library user) source files with the same line

```
#include <gnuastro/headername.h>
```

Note that the GSL convention for header file names is `gsl_specialname.h`, so your include directive for a GSL header must be something like `#include <gsl/gsl_specialname.h>`. Gnuastro doesn’t follow this GSL guideline because of the repeated `gsl` in the include directive. It can be confusing and cause bugs for beginners. All Gnuastro (and GSL) headers must be located within a unique directory and will not be mixed with other headers. Therefore the ‘`gsl_`’ prefix to the header file names is redundant\(^\text{18}\).

- All installed functions and variables should also include the base-name of the file in which they are defined as prefix, using underscores to separate words\(^\text{19}\). The same applies to exported macros, but in upper case. For example in Gnuastro’s top source directory, the prototype of function `gal_box_border_from_center` is in `lib/gnuastro/box.h`, and the macro `GAL_POLYGON_MAX_CORNERS` is defined in `lib/gnuastro/polygon.h`.

This is necessary to give any user (who is not familiar with the library structure) the ability to follow the code. This convention does make the function names longer (a little harder to write), but the extra documentation it provides plays an important role in Gnuastro and is worth the cost.

- There should be no trailing white space in a line. To do this automatically every time you save a file in Emacs, add the following line to your `~/.emacs` file.

```
(add-hook 'before-save-hook 'delete-trailing-whitespace)
```

---

\(^\text{17}\) [https://www.gnu.org/software/gsl/design/gsl-design.html\#SEC15](https://www.gnu.org/software/gsl/design/gsl-design.html\#SEC15)

\(^\text{18}\) For GSL, this prefix has an internal technical application: GSL’s architecture mixes installed and not-installed headers in the same directory. This prefix is used to identify their installation status. Therefore this filename prefix in GSL a technical internal issue (for developers, not users).

\(^\text{19}\) The convention to use underscores to separate words, called “snake case” (or “snake_case”). This is also recommended by the GNU coding standards.
• There should be no tabs in the indentation\footnote{If you use Emacs, Gnuastro’s `.dir-locals.el` file will automatically never use tabs for indentation. To make this a default in all your Emacs sessions, you can add the following line to your `~/.emacs` file: `(setq-default indent-tabs-mode nil)`.}

• Individual, contextually similar, functions in a source file are separated by 5 blank lines to be easily seen to be related in a group when parsing the source code by eye. In Emacs you can use \texttt{CTRL-u 5 CTRL-o}.

• One group of contextually similar functions in a source file is separated from another with 20 blank lines. In Emacs you can use \texttt{CTRL-u 20 CTRL-o}. Each group of functions has short descriptive title of the functions in that group. This title is surrounded by asterisks (*) to make it clearly distinguishable. Such contextual grouping and clear title are very important for easily understanding the code.

• Always read the comments before the patch of code under it. Similarly, try to add as many comments as you can regarding every patch of code. Effectively, we want someone to get a good feeling of the steps, without having to read the C code and only by reading the comments. This follows similar principles as Literate programming (\url{https://en.wikipedia.org/wiki/Literate_programming}).

The last two conventions are not common and might benefit from a short discussion here. With a good experience in advanced text editor operations, the last two are redundant for a professional developer. However, recall that Gnuastro aspires to be friendly to unfamiliar, and inexperienced (in programming) eyes. In other words, as discussed in Section 1.2 [Gnuastro manifesto: Science and its tools], page 2, we want the code to appear welcoming to someone who is completely new to coding (and text editors) and only has a scientific curiosity.

Newcomers to coding and development, who are curious enough to venture into the code, will probably not be using (or have any knowledge of) advanced text editors. They will see the raw code in the web page or on a simple text editor (like Gedit) as plain text. Trying to learn and understand a file with dense functions that are all spaced with one or two blank lines can be very taunting for a newcomer. But when they scroll through the file and see clear titles and meaningful spaces for similar functions, we are helping them find and focus on the part they are most interested in sooner and easier.
**GNU Emacs, the recommended text editor:** GNU Emacs is an extensible and easily customizable text editor which many programmers rely on for developing due to its countless features. Among them, it allows specification of certain settings that are applied to a single file or to all files in a directory and its sub-directories. In order to harmonize code coming from different contributors, Gnuastro comes with a `.dir-locals.el` file which automatically configures Emacs to satisfy most of the coding conventions above when you are using it within Gnuastro’s directories. Thus, Emacs users can readily start hacking into Gnuastro. If you are new to developing, we strongly recommend this editor. Emacs was the first project released by GNU and is still one of its flagship projects. Some resources can be found at:

**Official manual**
At [https://www.gnu.org/software/emacs/manual/emacs.html](https://www.gnu.org/software/emacs/manual/emacs.html). This is a great and very complete manual which is being improved for over 30 years and is the best starting point to learn it. It just requires a little patience and practice, but rest assured that you will be rewarded. If you install Emacs, you also have access to this manual on the command-line with the following command (see Section 4.3.4 [Info], page 174).

`$ info emacs`

**A guided tour of emacs**

**Unofficial mini-manual**

### 12.4 Program source

Besides the fact that all the programs share some functions that were explained in Chapter 11 [Library], page 513, everything else about each program is completely independent. Recall that Gnuastro is written for an active astronomer/scientist (not a passive one who just uses a software). It must thus be easily navigable. Hence there are fixed source files (that contain fixed operations) that must be present in all programs, these are discussed fully in Section 12.4.1 [Mandatory source code files], page 677. To easily understand the explanations in this section you can use Section 12.4.2 [The TEMPLATE program], page 680, which contains the bare minimum code for one working program. This template can also be used to easily add new utilities: just copy and paste the directory and change `TEMPLATE` with your program’s name.

### 12.4.1 Mandatory source code files

Some programs might need lots of source files and if there is no fixed convention, navigating them can become very hard for a new inquirer into the code. The following source files exist in every program’s source directory (which is located in `bin/progname`). For small programs, these files are enough. Larger programs will need more files and developers are encouraged to define any number of new files. It is just important that the following list of files exist and do what is described here. When creating other source files, please choose
filenames that are a complete single word: don’t abbreviate (abbreviations are cryptic). For a minimal program containing all these files, see Section 12.4.2 [The TEMPLATE program], page 680.

main.c Each executable has a main function, which is located in main.c. Therefore this file is the starting point when reading any program’s source code. No actual processing functions must be defined in this file, the function(s) in this file are only meant to connect the most high level steps of each program. Generally, main will first call the top user interface function to read user input and make all the preparations. Then it will pass control to the top processing function for that program. The functions to do both these jobs must be defined in other source files.

main.h All the major parameters which will be used in the program must be stored in a structure which is defined in main.h. The name of this structure is usually prognameparams, for example cropparams or noisechiselparams. So #include "main.h" will be a staple in all the source codes of the program. It is also regularly the first (and only) argument most of the program’s functions which greatly helps in readability.

Keeping all the major parameters of a program in this structure has the major benefit that most functions will only need one argument: a pointer to this structure. This will significantly facilitate the job of the programmer, the inquirer and the computer. All the programs in Gnuastro are designed to be low-level, small and independent parts, so this structure should not get too large.

The main root structure of all programs contains at least one instance of the gal_options_common_params structure. This structure will keep the values to all common options in Gnuastro’s programs (see Section 4.1.2 [Common options], page 156). This top root structure is conveniently called p (short for parameters) by all the functions in the programs and the common options parameters within it are called cp. With this convention any reader can immediately understand where to look for the definition of one parameter. For example you know that p->cp->output is in the common parameters while p->threshold is in the program’s parameters.

With this basic root structure, source code of functions can potentially become full of structure de-reference operators (->) which can make the code very unreadable. In order to avoid this, whenever a structure element is used more than a couple of times in a function, a variable of the same type and with the same name (so it can be searched) as the desired structure element should be defined with the value of the root structure inside of it in definition time. Here is an example.

    char *hdu=p->cp.hdu;
    float threshold=p->threshold;

args.h The options particular to each program are defined in this file. Each option is defined by a block of parameters in program_options. These blocks are all you should modify in this file, leave the bottom group of definitions untouched. These are fed directly into the GNU C library’s Argp facilities and it is recom-
mended to have a look at that for better understand what is going on, although this is not required here.

Each element of the block defining an option is described under `argp_option` in `bootstrapped/lib/argp.h` (from Gnuastro’s top source file). Note that the last few elements of this structure are Gnuastro additions (not documented in the standard Argp manual). The values to these last elements are defined in `lib/gnuastro/type.h` and `lib/gnuastro-internal/options.h` (from Gnuastro’s top source directory).

`ui.h` Besides declaring the exported functions of `ui.c`, this header also keeps the “key”s to every program-specific option. The first class of keys for the options that have a short-option version (single letter, see Section 4.1.1.2 [Options], page 154). The character that is defined here is the option’s short option name. The list of available alphabet characters can be seen in the comments. Recall that some common options also take some characters, for those, see `lib/gnuastro-internal/options.h`. The second group of options are those that don’t have a short option alternative. Only the first in this group needs a value (1000), the rest will be given a value by C’s `enum` definition, so the actual value is irrelevant and must never be used, always use the name.

`ui.c` Everything related to reading the user input arguments and options, checking the configuration files and checking the consistency of the input parameters before the actual processing is run should be done in this file. Since most functions are the same, with only the internal checks and structure parameters differing. We recommend going through the `ui.c` of Section 12.4.2 [The TEMPLATE program], page 680, or several other programs for a better understanding. The most high-level function in `ui.c` is named `ui_read_check_inputs_setup`. It accepts the raw command-line inputs and a pointer to the root structure for that program (see the explanation for `main.h`). This is the function that `main` calls. The basic idea of the functions in this file is that the processing functions should need a minimum number of such checks. With this convention an inquirer who only wants to understand only one part (mostly the processing part and not user input details and sanity checks) of the code can easily do so in the later files. It also makes all the errors related to input appear before the processing begins which is more convenient for the user.

`progname.c`, `progname.h` The high-level processing functions in each program are in a file named `progname.c`, for example `crop.c` or `noisechisel.c`. The function within these files which `main` calls is also named after the program, for example

```c
void
crop(struct cropparams *p)
```
or

```c
void
noisechisel(struct noisechiselparms *p)
```
In this manner, if an inquirer is interested the processing steps, they can immediately come and check this file for the first processing step without having to
go through `main.c` and `ui.c` first. In most situations, any failure in any step of the programs will result in an informative error message and an immediate abort in the program. So there is usually no need for return values. Under more complicated situations where a return value might be necessary, `void` will be replaced with an `int` in the examples above. This value must be directly returned by `main`, so it has to be an `int`.

**authors-cite.h**

This header file keeps the global variable for the program authors and its BibTeX record for citation. They are used in the outputs of the common options `--version` and `--cite`, see Section 4.1.2.3 [Operating mode options], page 162.

**progname-complete.bash**

This shell script is used for implementing auto-completion features when running Gnuastro’s programs within GNU Bash. For more on the concept of shell auto-completion and how it is managed in Gnuastro, see Section 12.8 [Bash programmable completion], page 685.

These files assume a set of common shell functions that have the prefix `_gnuastro_autocomplete_` in their name and are defined in `bin/complete.bash.in` (of the source directory, and under version control) and `bin/complete.bash.built` (built during the building of Gnuastro in the build directory). During Gnuastro’s build, all these Bash completion files are merged into one file that is installed and the user can `source` them into their Bash startup file, for example see Section 1.1 [Quick start], page 1.

### 12.4.2 The TEMPLATE program

The extra creativity offered by libraries comes at a cost: you have to actually write your `main` function and get your hands dirty in managing user inputs: are all the necessary parameters given a value? is the input in the correct format? do the options and the inputs correspond? and many other similar checks. So when an operation has well-defined inputs and outputs and is commonly needed, it is much more worthwhile to simply do use all the great features that Gnuastro has already defined for such operations.

To make it easier to learn/apply the internal program infra-structure discussed in Section 12.4.1 [Mandatory source code files], page 677, in the Section 3.2.2 [Version controlled source], page 132, Gnuastro ships with a template program . This template program is not available in the Gnuastro tarball so it doesn’t confuse people using the tarball. The `bin/TEMPLATE` directory in Gnuastro’s Git repository contains the bare-minimum files necessary to define a new program and all the basic/necessary files/functions are pre-defined there.

Below you can see a list of initial steps to take for customizing this template. We just assume that after cloning Gnuastro’s history, you have already bootstrapped Gnuastro, if not, please see Section 3.2.2.1 [Bootstrapping], page 133.

1. Select a name for your new program (for example `myprog`).
2. Copy the `TEMPLATE` directory to a directory with your program’s name:
   ```bash
   $ cp -R bin/TEMPLATE bin/myprog
   ```
3. As with all source files in Gnuastro, all the files in template also have a copyright notice at their top. Open all the files and correct these notices: 1) The first line
contains a single-line description of the program. 2) In the second line only the name
or your program needs to be fixed and 3) Add your name and email as a “Contributing
author”. As your program grows, you will need to add new files, don’t forget to add
this notice in those new files too, just put your name and email under “Original author”
and correct the copyright years.

4. Open configure.ac in the top Gnuastro source. This file manages the operations that
are done when a user runs ./configure. Going down the file, you will notice repetitive
parts for each program. You will notice that the program names follow an alphabetic
ordering in each part. There is also a commented line/patch for the TEMPLATE program
in each part. You can copy one line/patch (from the program above or below your
desired name for example) and paste it in the proper place for your new program.
Then correct the names of the copied program to your new program name. There are
multiple places where this has to be done, so be patient and go down to the bottom
of the file. Ultimately add bin/myprog/Makefile to AC_CONFIG_FILES, only here the
ordering depends on the length of the name (it isn’t alphabetical).

5. Open Makefile.am in the top Gnuastro source. Similar to the previous step, add your
new program similar to all the other programs. Here there are only two places: 1) at
the top where we define the conditionals (three lines per program), and 2) immediately
under it as part of the value for SUBDIRS.

6. Open doc/Makefile.am and similar to Makefile.am (above), add the proper entries
for the man-page of your program to be created (here, the variable that keeps all the
man-pages to be created is dist_man_MANS). Then scroll down and add a rule to build
the man-page similar to the other existing rules (in alphabetical order). Don’t forget
to add a short one-line description here, it will be displayed on top of the man-page.

7. Change TEMPLATE.c and TEMPLATE.h to myprog.c and myprog.h in the file names:

    $ cd bin/myprog
    $ mv TEMPLATE.c myprog.c
    $ mv TEMPLATE.h myprog.h

8. Correct all occurrences of TEMPLATE in the input files to myprog (in short or long
format). You can get a list of all occurrences with the following command. If you
use Emacs, it will be able to parse the Grep output and open the proper file and line
automatically. So this step can be very easy.

    $ grep --color -nHi -e template *

9. Run the following commands to re-build the configuration and build system, and then
to configure and build Gnuastro (which now includes your exciting new program).

    $ autoreconf -f
    $ ./configure
    $ make

10. You are done! You can now start customizing your new program to do your special
processing. When its complete, just don’t forget to add checks also, so it can be
tested at least once on a user’s system with make check, see Section 12.7 [Test scripts],
page 684. Finally, if you would like to share it with all Gnuastro users, inform us so
we merge it into Gnuastro’s main history.
12.5 Documentation

Documentation (this book) is an integral part of Gnuastro (see Section 1.2 [Gnuastro manifesto: Science and its tools], page 2). Documentation is not considered a separate project and must be written by its developers. Users can make edits/corrections, but the initial writing must be by the developer. So, no change is considered valid for implementation unless the respective parts of the book have also been updated. The following procedure can be a good suggestion to take when you have a new idea and are about to start implementing it.

The steps below are not a requirement, the important thing is that when you send your work to be included in Gnuastro, the book and the code have to both be fully up-to-date and compatible, with the purpose of the update very clearly explained. You can follow any strategy you like, the following strategy was what we have found to be most useful until now.

1. Edit the book and fully explain your desired change, such that your idea is completely embedded in the general context of the book with no sense of discontinuity for a first time reader. This will allow you to plan the idea much more accurately and in the general context of Gnuastro (a particular program or library). Later on, when you are coding, this general context will significantly help you as a road-map.

A very important part of this process is the program/library introduction. These first few paragraphs explain the purposes of the program or library and are fundamental to Gnuastro. Before actually starting to code, explain your idea’s purpose thoroughly in the start of the respective/new section you wish to work on. While actually writing its purpose for a new reader, you will probably get some valuable and interesting ideas that you hadn’t thought of before. This has occurred several times during the creation of Gnuastro.

If an introduction already exists, embed or blend your idea’s purpose with the existing introduction. We emphasize that doing this is equally useful for you (as the programmer) as it is useful for the user (reader). Recall that the purpose of a program is very important, see Section 12.2 [Program design philosophy], page 672.

As you have already noticed for every program/library, it is very important that the basics of the science and technique be explained in separate subsections prior to the ‘Invoking Programname’ subsection. If you are writing a new program or your addition to an existing program involves a new concept, also include such subsections and explain the concepts so a person completely unfamiliar with the concepts can get a general initial understanding. You don’t have to go deep into the details, just enough to get an interested person (with absolutely no background) started with some good pointers/links to where they can continue studying if they are more interested. If you feel you can’t do that, then you have probably not understood the concept yourself. If you feel you don’t have the time, then think about yourself as the reader in one year: you will forget almost all the details, so now that you have done all the theoretical preparations, add a few more hours and document it. Therefore in one year, when you find a bug or want to add a new feature, you don’t have to prepare as much. Have in mind that your only limitation in length is the fatigue of the reader after reading a long text, nothing else. So as long as you keep it relevant/interesting for the reader, there is no page number limit/cost.
Chapter 12: Developing

It might also help if you start discussing the usage of your idea in the ‘Invoking ProgramName’ subsection (explaining the options and arguments you have in mind) at this stage too. Actually starting to write it here will really help you later when you are coding.

2. After you have finished adding your initial intended plan to the book, then start coding your change or new program within the Gnuastro source files. While you are coding, you will notice that somethings should be different from what you wrote in the book (your initial plan). So correct them as you are actually coding, but don’t worry too much about missing a few things (see the next step).

3. After your work has been fully implemented, read the section documentation from the start and see if you didn’t miss any change in the coding and to see if the context is fairly continuous for a first time reader (who hasn’t seen the book or had known Gnuastro before you made your change).

4. If the change is notable, also update the NEWS file.

12.6 Building and debugging

To build the various programs and libraries in Gnuastro, the GNU build system is used which defines the steps in Section 1.1 [Quick start], page 1. It consists of GNU Autoconf, GNU Automake and GNU Libtool which are collectively known as GNU Autotools. They provide a very portable system to check the hosts environment and compile Gnuastro based on that. They also make installing everything in their standard places very easy for the programmer.

Most of the small caps files that you see in the top source directory of the tarball are created by these three tools (see Section 3.2.2 [Version controlled source], page 132). To facilitate the building and testing of your work during development, Gnuastro comes with two useful scripts:

**developer-build**

This is more fully described in Section 3.3.1.4 [Configure and build in RAM], page 144. During development, you will usually run this command only once (at the start of your work).

**tests/during-dev.sh**

This script is designed to be run each time you make a change and want to test your work (with some possible input and output). The script itself is heavily commented and thoroughly describes the best way to use it, so we won’t repeat it here.

As a short summary: you specify the build directory, an output directory (for the built program to be run in, and also contains the inputs), the program’s short name and the arguments and options that it should be run with. This script will then build Gnuastro, go to the output directory and run the built executable from there. One option for the output directory might be your desktop, so you can easily see the output files and delete them when you are finished. The main purpose of these scripts is to keep your source directory clean and facilitate your development.

By default all the programs are compiled with optimization flags for increased speed. A side effect of optimization is that valuable debugging information is lost. All the libraries
are also linked as shared libraries by default. Shared libraries further complicate the debugging process and significantly slow down the compilation (the make command). So during development it is recommended to configure Gnuastro as follows:

```bash
$ ./configure --enable-debug
```

In developer-build you can ask for this behavior through the `--debug` option, see Section 3.3.2 [Separate build and source directories], page 145.

In order to understand the building process, you can go through the Autoconf, Automake and Libtool manuals, like all GNU manuals they provide both a great tutorial and technical documentation. The “A small Hello World” section in Automake’s manual (in chapter 2) can be a good starting guide after you have read the separate introductions.

### 12.7 Test scripts

As explained in Section 3.3.3 [Tests], page 148, for every program some simple tests are written to check the various independent features of the program. All the tests are placed in the `tests/` directory. The `tests/prepconf.sh` script is the first ‘test’ that will be run. It will copy all the configuration files from the various directories to a `tests/.gnuastro` directory (which it will make) so the various tests can set the default values. This script will also make sure the programs don’t go searching for user and system wide configuration files to avoid the mixing of values with different Gnuastro version on the system.

For each program, the tests are placed inside directories with the program name. Each test is written as a shell script. The last line of this script is the test which runs the program with certain parameters. The return value of this script determines the fate of the test, see the “Support for test suites” chapter of the Automake manual for a very nice and complete explanation. In every script, two variables are defined at first: `prog` and `execname`. The first specifies the program name and the second the location of the executable.

The most important thing to have in mind about all the test scripts is that they are run from inside the `tests/` directory in the “build tree”. Which can be different from the directory they are stored in (known as the “source tree”)\(^\text{21}\). This distinction is made by GNU Autoconf and Automake (which configure, build and install Gnuastro) so that you can install the program even if you don’t have write access to the directory keeping the source files. See the “Parallel build trees (a.k.a VPATH builds)” in the Automake manual for a nice explanation.

Because of this, any necessary inputs that are distributed in the tarball\(^\text{22}\), for example the catalogs necessary for checks in MakeProfiles and Crop, must be identified with the `$topsrc` prefix instead of `../` (for the top source directory that is unpacked). This `$topsrc` variable points to the source tree where the script can find the source data (it is defined in `tests/Makefile.am`). The executables and other test products were built in the build tree (where they are being run), so they don’t need to be prefixed with that variable. This is also true for images or files that were produced by other tests.

---

\(^\text{21}\) The `developer-build` script also uses this feature to keep the source and build directories separate (see Section 3.3.2 [Separate build and source directories], page 145).

\(^\text{22}\) In many cases, the inputs of a test are outputs of previous tests, this doesn’t apply to this class of inputs. Because all outputs of previous tests are in the “build tree”.
Under development: While work on TAB completion is ongoing, it is not yet fully ready, please see the notice at the start of Section 4.1.3 [Shell TAB completion (highly customized)], page 166.

Gnuastro provides Programmable completion facilities in Bash. This greatly helps users reach their desired result with minimal keystrokes, and helps them spend less time on figuring out the option names and values their acceptable values. Gnuastro's completion script not only completes the half-written commands, but also prints suggestions based on previous arguments.

Imagine a scenario where we need to download three columns containing the right ascension, declination, and parallax from the GAIA EDR3 dataset. We have to make sure how these columns are abbreviated or spelled. So we can call the command below, and store the column names in a file such as `gaia-edr3-columns.txt`.

```bash
$ astquery gaia --information > gaia-edr3-columns.txt
```

Then we need to memorize or copy the column names of interest, and specify an output fits file name such as `gaia.fits`:

```bash
$ astquery gaia --dataset=edr3 --output=gaia.fits --column=ra,dec,parallax
```

However, this is much easier using the auto-completion feature:

```bash
$ astquery gaia --dataset=edr3 --output=gaia.fits --column=[TAB]
```

After pressing `[TAB]`, a full list of gaia edr3 dataset column names will be displayed. Typing the first key of the desired column and pressing `[TAB]` again will limit the displayed list to only the matching ones until the desired column is found.

12.8.1 Bash TAB completion tutorial

When a user presses the `[TAB]` key while typing commands, Bash will inspect the input to find a relevant “completion specification”, or `compspec`. If available, the `compspec` will generate a list of possible suggestions to complete the current word. A custom `compspec` can be generated for any command using `bash completion builtins`\(^{23}\) and the bash variables that start with the `COMP` keyword\(^{24}\).

First, let’s see a quick example of how you can make a completion script in just one line of code. With the command below, we are asking Bash to give us three suggestions for `echo`: `foo`, `bar` and `bAr`. Please run it in your terminal for the next steps.

```bash
$ complete -W "foo bar bAr" echo
```

The possible completion suggestions are fed into `complete` using the `-W` option followed by a list of space delimited words. Let’s see it in action:

```bash
$ echo [TAB][TAB]
bar bAr foo
```

---


Nicely done! Just note that the strings are sorted alphabetically, not in the original order. Also, an arbitrary number of space characters are printed between them (based on the number of suggestions and terminal size and etc). Now, if you type ‘f’ and press [TAB], bash will automatically figure out that you wanted foo and it be completed right away:

```
$ myprogram f[TAB]
$ myprogram foo
```

However, nothing will happen if you type ‘b’ and press [TAB] only once. This is because of the ambiguity: there isn’t enough information to figure out which suggestion you want: bar or bAr? So, if you press [TAB] twice, it will print out all the options that start with ‘b’:

```
$ echo b[TAB][TAB]
bar bAr
$ echo ba[TAB]
$ echo bar
```

Not bad for a simple program. But what if you need more control? By passing the -F option to complete instead of -W, it will run a function for generating the suggestions, instead of using a static string. For example, let’s assume that the expected value after foo is the number of files in the current directory. Since the logic is getting more complex, let’s write and save the commands below into a shell script with an arbitrary name such as completion-tutorial.sh:

```
$ cat completion-tutorial.sh
_echo(){
    if [ "$3" == "foo" ]; then
        COMPREPLY=( $(ls | wc -l) )
    else
        COMPREPLY=( $(compgen -W "foo bar bAr" -- "$2") )
    fi
}
complete -F _echo echo
```

We will look at it in detail soon. But for now, let’s source the file into your current terminal and check if it works as expected:

```
$ source completion-tutorial.sh
$ echo [TAB][TAB]
foo bar bAr
$ echo foo [TAB]
$ touch empty.txt
$ echo foo [TAB]
```

Success! As you see, this allows for setting up highly customized completion scripts. Now let’s have a closer look at the completion-tutorial.sh completion script from above. First, the ‘-F’ option in front the complete command indicates that we want shell to execute the _echo function whenever echo is called. As a convention, the function name should be the same as the program name, but prefixed with an underscore (‘_’).

Within the _echo function, we’re checking if $3 is equal to foo. In Bash’s auto-complete, $3 means the word before current cursor position. In fact, these are the arguments that the _echo function is receiving:
The name of the command, here it is `echo'.

The current word being completed (empty unless we are in the middle of typing a word).

The word before the word being completed.

To tell the completion script what to reply with, we use the `COMPREPLY` array. This array holds all the suggestions that `complete` will show for the user in the end. In the example above, we simply give it the string output of `ls | wc -l`.

Finally, we have the `compgen` command. According to bash programmable completion builtins manual, the command `compgen [OPTION] [WORD]` generates possible completion matches for [WORD] according to [OPTIONS]. Using the `-W` option asks `compgen` to generate a list of words from an input string. This is known as *Word Splitting*\(^\text{25}\). `compgen` will automatically use the `$IFS` variable to split the string into a list of words. You can check the default delimiters by calling:

```bash
printf %q "$IFS"
```

The default value of `$IFS` might be `\t\n`. This means the SPACE, TAB, and New-line characters. Finally, notice the `'-- "$2"'` in this command:

```bash
COMPREPLY=( $(compgen -W "foo bar bAr" -- "$2") )
```

Here, the `'--'` instructs `compgen` to only reply with a list of words that match `$2`, i.e. the current word being completed. That is why when you type the letter ‘b’, `complete` will reply only with its matches (`‘bar’` and `‘bAr’`), and will exclude `‘foo’`.

Let’s get a little more realistic, and develop a very basic completion script for one of Gnuastro’s programs. Since the `--help` option will list all the options available in Gnuastro’s programs, we are going to use its output and create a very basic TAB completion for it. Note that the actual TAB completion in Gnuastro is a little more complex than this and fully described in Section 12.8.2 [Implementing TAB completion in Gnuastro], page 689. But this is a good exercise to get started.

We’ll use `asttable` as the demo, and the goal is to suggest all options that this program has to offer. You can print all of them (with a lot of extra information) with this command:

```bash
$ asttable --help
```

Let’s write an `awk` script that prints all of the long options. When printing the option names we can safely ignore the short options because if a user knows about the short options, s/he already knows exactly what they want! Also, due to their single-character length, they will be too cryptic without their descriptions.

One way to catch the long options is through `awk` as shown below. We only keep the lines that 1) starting with an empty space, 2) their first no-white character is ‘-’ and that have the format of ‘-‘ followed by any number of numbers or characters. Within those lines, if the first word ends in a comma (‘,‘), the first word is the short option, so we want the second word (which is the long option). Otherwise, the first word is the long option. But for options that take a value, this will also include the format of the value (for example `--column=STR`). So with a `sed` command, we remove everything that is after the equal sign, but keep the equal sign itself (to highlight to the user that this option should have a value).

```bash
$ asttable --help \
```

If we wanted to show all the options to the user, we could simply feed the values of the command above to `compgen` and `COMPREPLY` subsequently. But, we need smarter completions: we want to offer suggestions based on the previous options that have already been typed in. Just Beware! Sometimes the program might not be acting as you expected. In that case, using debug messages can clear things up. You can add a `echo` command before the completion function ends, and check all current variables. This can save a lot of headaches, since things can get complex.

Take the option `--wcsfile=` for example. This option accepts a FITS file. Usually, the user is trying to feed a FITS file from the current directory. So it would be nice if we could help them and print only a list of FITS files sitting in the current directory – or whatever directory they have typed-in so far.

But there’s a catch. When splitting the user’s input line, Bash will consider `=` as a separate word. To avoid getting caught in changing the `IFS` or `WORDBREAKS` values, we will simply check for `=` and act accordingly. That is, if the previous word is a `=`, we’ll ignore it and take the word before that as the previous word. Also, if the current word is a `=`, ignore it completely. Taking all of that into consideration, the code below might serve well:

```bash
_asttable(){
    if [ "$2" = "=" ]; then word=""
    else
        word="$2"
    fi

    if [ "$3" = "=" ]; then prev="${COMP_WORDS[COMP_CWORD-2]}"
    else
        prev="${COMP_WORDS[COMP_CWORD-1]}"
    fi

    case "$prev" in
      --wcsfile)
        COMPREPLY=( $(compgen -f -X "!*.[fF][iI][tT][sS]" -- "$word") )
        ;;
      esac
}
complete -o nospace -F _asttable asttable
```

To test the code above, write it into `asttable-tutorial.sh`, and load it into your running terminal with this command:

```
$ source asttable-tutorial.sh
```

If you then go to a directory that has at least one FITS file (with a `.fits` suffix, among other files), you can checkout the function by typing the following command. You will see that only files ending in `.fits` are shown, not any other file.

```
asttable --wcsfile=[TAB][TAB]
```
The code above first identifies the current and previous words. It then checks if the previous word is equal to `--wcsfile` and if so, fills `COMPREPLY` array with the necessary suggestions. We are using `case` here (instead of `if`) because in a real scenario, we need to check many more values and `case` is far better suited for such cases (cleaner and more efficient code).

The `-f` option in `compgen` indicates we’re looking for a file. The `-X` option filters out the filenames that match the next regular expression pattern. Therefore we should start the regular expression with `!` if we want the files matching the regular expression. The `-- "$word"` component collects only filenames that match the current word being typed. And last but not least, the ‘-o nospace’ option in the `complete` command instructs the completion script to not append a white space after each suggestion. That is important because the long format of an option, its value is more clear when it sticks to the option name with a ‘=’ sign.

You have now written a very basic and working TAB completion script that can easily be generalized to include more options (and be good for a single/simple program). However, Gnuastro has many programs that share many similar things and the options are not independent. Also, complex situations do often come up: for example some people use a `.fit` suffix for FITS files and others don’t even use a suffix at all! So in practice, things need to get a little more complicated, but the core concept is what you learnt in this section. We just modularize the process (breaking logically independent steps into separate functions to use in different situations). In Section 12.8.2 [Implementing TAB completion in Gnuastro], page 689, we will review the generalities of Gnuastro’s implementation of Bash TAB completion.

### 12.8.2 Implementing TAB completion in Gnuastro

The basics of Bash auto-completion was reviewed in Section 12.8.1 [Bash TAB completion tutorial], page 685. Gnuastro is a very complex package of many programs, that have many similar features, so implementing those principles in an easy to maintain manner requires a modular solution. As a result, Bash’s TAB completion is implemented as multiple files in Gnuastro:

- **bin/completion.bash.built** (in build directory, automatically created)
  This file contains the values of all Gnuastro options or arguments that take fixed strings as values (not file names). For example the names of Arithmetic’s operators (see Section 6.2.3 [Arithmetic operators], page 276), or spectral line names (like `--obsline` in Section 9.1.3.1 [CosmicCalculator input options], page 475).
  This file is created automatically during the building of Gnuastro. The recipe to build it is available in Gnuastro’s top-level `Makefile.am` (under the target `bin/completion.bash`). It parses the respective Gnuastro source file that contains the necessary user-specified strings. All the acceptable values values are then stored as shell variables (within a function).

- **bin/completion.bash.in** (in source directory, under version control)
  All the low-level completion functions that are common to all programs are stored here. It thus contains functions that will parse the command-line or files, or suggest the completion replies.
PROGNAME-complete.bash (in source directory, under version control)

All Gnuastro programs contain a PROGNAME-complete.bash script within their
source (for more on the fixed files of each program, see Section 12.4 [Program
source], page 677). This file contains the very high-level (program-specific) Bash
programmable completion features that are almost always defined in Gnuastro-
generic Bash completion file (bin/completion.bash.in).

The top-level function that is called by Bash should be called _gnuastro_
autocomplete_PROGNAME and its last line should be the complete command
of Bash which calls this function. The contents of _gnuastro_autocomplete_
PROGNAME are almost identical for all the programs, it is just a very high-level
function that either calls _gnuastro_autocomplete_PROGNAME_arguments to
manage suggestions for the program’s arguments or _gnuastro_autocomplete_
PROGNAME_option_value to manage suggestions for the program’s option val-
ues.

The scripts above follow the following conventions. After reviewing the list, please also look
into the functions for examples of each point.

- No global shell variables in any completion script: the contents of the files above are
directly loaded into the user’s environment. So to keep the user’s environment clean
and avoid annoyance to the users, everything should be defined as shell functions, and
any variable within the functions should be set as local.
- All the function names should start with `_gnuastro_autocomplete_`, again to avoid
populating the user’s function name-space with possibly conflicting names.
- Outputs of functions should be written in the local variables of the higher-level func-
tions that called them.

12.9 Developer’s checklist

This is a checklist of things to do after applying your changes/additions in Gnuastro:

1. If the change is non-trivial, write test(s) in the tests/progname/ directory to
test the change(s)/addition(s) you have made. Then add their file names to
tests/Makefile.am.

2. If your change involves a change in command-line behavior of a Gnuastro program
or script (for example, adding a new option or argument), create or update the respec-
tive bin/PROGNAME/completion.sh file described under the Section 12.8 [Bash
programmable completion], page 685, section.

3. Run `$ make check` to make sure everything is working correctly.

4. Make sure the documentation (this book) is completely up to date with your changes,
see Section 12.5 [Documentation], page 682.

5. Commit the change to your issue branch (see Section 12.12.3 [Production workflow],
page 696, and Section 12.12.4 [Forking tutorial], page 697). Afterwards, run Autoreconf
to generate the appropriate version number:

    `$ autoreconf -f`

6. Finally, to make sure everything will be built, installed and checked correctly run the
following command (after re-configuring, and re-building). To greatly speed up the
process, use multiple threads (8 in the example below, change it appropriately)
$ make distcheck -j8

This command will create a distribution file (ending with .tar.gz) and try to compile it in the most general cases, then it will run the tests on what it has built in its own mini-environment. If $ make distcheck finishes successfully, then you are safe to send your changes to us to implement or for your own purposes. See Section 12.12.3 [Production workflow], page 696, and Section 12.12.4 [Forking tutorial], page 697.

12.10 Gnuastro project webpage

Gnuastro’s central management hub (https://savannah.gnu.org/projects/gnuastro/)\(^{26}\) is located on GNU Savannah (https://savannah.gnu.org/)\(^{27}\). Savannah is the central software development management system for many GNU projects. Through this central hub, you can view the list of activities that the developers are engaged in, their activity on the version controlled source, and other things. Each defined activity in the development cycle is known as an ‘issue’ (or ‘item’). An issue can be a bug (see Section 1.8 [Report a bug], page 11), or a suggested feature (see Section 1.9 [Suggest new feature], page 13) or an enhancement or generally any one job that is to be done. In Savannah, issues are classified into three categories or ‘tracker’s:

**Support** This tracker is a way that (possibly anonymous) users can get in touch with the Gnuastro developers. It is a complement to the bug-gnuastro mailing list (see Section 1.8 [Report a bug], page 11). Anyone can post an issue to this tracker. The developers will not submit an issue to this list. They will only reassign the issues in this list to the other two trackers if they are valid\(^ {28}\). Ideally (when the developers have time to put on Gnuastro, please don’t forget that Gnuastro is a volunteer effort), there should be no open items in this tracker.

**Bugs** This tracker contains all the known bugs in Gnuastro (problems with the existing tools).

**Tasks** The items in this tracker contain the future plans (or new features/capabilities) that are to be added to Gnuastro.

All the trackers can be browsed by a (possibly anonymous) visitor, but to edit and comment on the Bugs and Tasks trackers, you have to be a registered on Savannah. When posting an issue to a tracker, it is very important to choose the ‘Category’ and ‘Item Group’ options accurately. The first contains a list of all Gnuastro’s programs along with ‘Installation’, ‘New program’ and ‘Webpage’. The “Item Group” contains the nature of the issue, for example if it is a ‘Crash’ in the software (a bug), or a problem in the documentation (also a bug) or a feature request or an enhancement.

The set of horizontal links on the top of the page (Starting with ‘Main’ and ‘Homepage’ and finishing with ‘News’) are the easiest way to access these trackers (and other major aspects of the project) from any part of the project web page. Hovering your mouse over them will open a drop down menu that will link you to the different things you can do on each tracker (for example, ‘Submit new’ or ‘Browse’). When you browse each tracker, you

---

\(^{26}\) https://savannah.gnu.org/projects/gnuastro/
\(^{27}\) https://savannah.gnu.org/
\(^{28}\) Some of the issues registered here might be due to a mistake on the user’s side, not an actual bug in the program.
can use the “Display Criteria” link above the list to limit the displayed issues to what you are interested in. The ‘Category’ and ‘Group Item’ (explained above) are a good starting point.

Any new issue that is submitted to any of the trackers, or any comments that are posted for an issue, is directly forwarded to the gnuastro-devel mailing list (https://lists.gnu.org/mailman/listinfo/gnuastro-devel, see Section 12.11 [Developing mailing lists], page 692, for more). This will allow anyone interested to be up to date on the over-all development activity in Gnuastro and will also provide an alternative (to Savannah) archiving for the development discussions. Therefore, it is not recommended to directly post an email to this mailing list, but do all the activities (for example add new issues, or comment on existing ones) on Savannah.

Do I need to be a member in Savannah to contribute to Gnuastro? No.

The full version controlled history of Gnuastro is available for anonymous download or cloning. See Section 12.12.3 [Production workflow], page 696, for a description of Gnuastro’s Integration-Manager Workflow. In short, you can either send in patches, or make your own fork. If you choose the latter, you can push your changes to your own fork and inform us. We will then pull your changes and merge them into the main project. Please see Section 12.12.4 [Forking tutorial], page 697, for a tutorial.

12.11 Developing mailing lists

To keep the developers and interested users up to date with the activity and discussions within Gnuastro, there are two mailing lists which you can subscribe to:

gnuastro-devel@gnu.org
(at https://lists.gnu.org/mailman/listinfo/gnuastro-devel)

All the posts made in the support, bugs and tasks discussions of Section 12.10 [Gnuastro project webpage], page 691, are also sent to this mailing address and archived. By subscribing to this list you can stay up to date with the discussions that are going on between the developers before, during and (possibly) after working on an issue. All discussions are either in the context of bugs or tasks which are done on Savannah and circulated to all interested people through this mailing list. Therefore it is not recommended to post anything directly to this mailing list. Any mail that is sent to it from Savannah to this list has a link under the title “Reply to this item at:”. That link will take you directly to the issue discussion page, where you can read the discussion history or join it.

While you are posting comments on the Savannah issues, be sure to update the meta-data. For example if the task/bug is not assigned to anyone and you would like to take it, change the “Assigned to” box, or if you want to report that it has been applied, change the status and so on. All these changes will also be circulated with the email very clearly.

gnuastro-commits@gnu.org
(at https://lists.gnu.org/mailman/listinfo/gnuastro-commits)

This mailing list is defined to circulate all commits that are done in Gnuastro’s version controlled source, see Section 3.2.2 [Version controlled source], page 132.
If you have any ideas, or suggestions on the commits, please use the bug and task trackers on Savannah to followup the discussion, do not post to this list. All the commits that are made for an already defined issue or task will state the respective ID so you can find it easily.

12.12 Contributing to Gnuastro

You have this great idea or have found a good fix to a problem which you would like to implement in Gnuastro. You have also become familiar with the general design of Gnuastro in the previous sections of this chapter (see Chapter 12 [Developing], page 670) and want to start working on and sharing your new addition/change with the whole community as part of the official release. This is great and your contribution is most welcome. This section and the next (see Section 12.9 [Developer’s checklist], page 690) are written in the hope of making it as easy as possible for you to share your great idea with the community.

In this section we discuss the final steps you have to take: legal and technical. From the legal perspective, the copyright of any work you do on Gnuastro has to be assigned to the Free Software Foundation (FSF) and the GNU operating system, or you have to sign a disclaimer. We do this to ensure that Gnuastro can remain free in the future, see Section 12.12.1 [Copyright assignment], page 693. From the technical point of view, in this section we also discuss commit guidelines (Section 12.12.2 [Commit guidelines], page 694) and the general version control workflow of Gnuastro in Section 12.12.3 [Production workflow], page 696, along with a tutorial in Section 12.12.4 [Forking tutorial], page 697.

Recall that before starting the work on your idea, be sure to checkout the bugs and tasks trackers in Section 12.10 [Gnuastro project webpage], page 691, and announce your work there so you don’t end up spending time on something others have already worked on, and also to attract similarly interested developers to help you.

12.12.1 Copyright assignment

Gnuastro’s copyright is owned by the Free Software Foundation (FSF) to ensure that Gnuastro always remains free. The FSF has also provided a Contributor FAQ (https://www.fsf.org/licensing/contributor-faq) to further clarify the reasons, so we encourage you to read it. Professor Eben Moglen, of the Columbia University Law School has given a nice summary of the reasons for this at https://www.gnu.org/licenses/why-assign. Below we are copying it verbatim for self consistency (in case you are offline or reading in print).

Under US copyright law, which is the law under which most free software programs have historically been first published, there are very substantial procedural advantages to registration of copyright. And despite the broad right of distribution conveyed by the GPL, enforcement of copyright is generally not possible for distributors: only the copyright holder or someone having assignment of the copyright can enforce the license. If there are multiple authors of a copyrighted work, successful enforcement depends on having the cooperation of all authors.

In order to make sure that all of our copyrights can meet the record keeping and other requirements of registration, and in order to be able to enforce the GPL most effectively, FSF requires that each author of code incorporated in FSF projects provide a copyright assignment, and, where appropriate, a disclaimer
of any work-for-hire ownership claims by the programmer’s employer. That way we can be sure that all the code in FSF projects is free code, whose freedom we can most effectively protect, and therefore on which other developers can completely rely.

Please get in touch with the Gnuastro maintainer (currently Mohammad Akhlaghi, mohammad -at- akhlaghi -dot- org) to follow the procedures. It is possible to do this for each change (good for a single contribution), and also more generally for all the changes/additions you do in the future within Gnuastro. So if you have already assigned the copyright of your work on another GNU software to the FSF, it should be done again for Gnuastro. The FSF has staff working on these legal issues and the maintainer will get you in touch with them to do the paperwork. The maintainer will just be informed in the end so your contributions can be merged within the Gnuastro source code.

Gnuastro will gratefully acknowledge (see Section 1.12 [Acknowledgments], page 15) all the people who have assigned their copyright to the FSF and have thus helped to guarantee the freedom and reliability of Gnuastro. The Free Software Foundation will also acknowledge your copyright contributions in the Free Software Supporter: https://www.fsf.org/free-software-supporter which will circulate to a very large community (225,910 people in July 2021). See the archives for some examples and subscribe to receive interesting updates. The very active code contributors (or developers) will also be recognized as project members on the Gnuastro project web page (see Section 12.10 [Gnuastro project webpage], page 691) and can be given a gnu.org email address. So your very valuable contribution and copyright assignment will not be forgotten and is highly appreciated by a very large community. If you are reluctant to sign an assignment, a disclaimer is also acceptable.

Do I need a disclaimer from my university or employer? It depends on the contract with your university or employer. From the FSF’s /gd/gnuorg/conditions.text: “If you are employed to do programming, or have made an agreement with your employer that says it owns programs you write, we need a signed piece of paper from your employer disclaiming rights to” Gnuastro. The FSF’s copyright clerk will kindly help you decide, please consult the following email address: “assign -at- gnu -dot- org”.

12.12.2 Commit guidelines

To be able to cleanly integrate your work with the other developers, never commit on the master branch (see Section 12.12.3 [Production workflow], page 696, for a complete discussion and Section 12.12.4 [Forking tutorial], page 697, for a cookbook example). In short, leave master only for changes you fetch, or pull from the official repository (see Section 3.2.2.2 [Synchronizing], page 134).

In the Gnuastro commit messages, we strive to follow these standards. Note that in the early phases of Gnuastro’s development, we are experimenting and so if you notice earlier commits don’t satisfy some of the guidelines below, it is because they predate that guideline.

Commit title

The commits have to start with one short descriptive title. The title is separated from the body with one blank line. Run git log to see some of the most recent commit messages as an example. In general, the title should satisfy the following conditions:
• It is best for the title to be short, about 60 (or even 50) characters. Most emulated command-line terminals are about 80 characters wide. However, we should also allow for the commit hashes which are printed in `git log --oneline`, and also branch names or the graph structure outputs of `git log` which are also commonly used.

• The title should not finish with any full-stops or periods (`.`).

Commit body

The body of the commit message is separated from the title by one empty line. Recall that anyone who has subscribed to `gnuastro-commits` mailing list will get the commit in their email after it has been pushed to `master`. People will also read them when they synchronize with the main Gnuastro repository (see Section 3.2.2.2 [Synchronizing], page 134). Finally, the commit messages will later be used to update the `NEWS` file on each release. Therefore the commit message body plays a very important role in the development of Gnuastro, so please adhere to the following guidelines.

• The body should be very descriptive. Start the commit message body by explaining what changes your commit makes from a user’s perspective (added, changed, or removed options, or arguments to programs or libraries, or modified algorithms, or new installation step, etc).

• Try to explain the committed contents as best as you can. Recall that the readers of your commit message do not necessarily have your current background. After some time you will also forget the context, so this request is not just for others. Therefore be very descriptive and explain as much as possible: what the bug/task was, justify the way you fixed it and discuss other possible solutions that you might not have included. For the last item, it is best to discuss them thoroughly as comments in the appropriate section of the code, but only give a short summary in the commit message. Note that all added and removed source code lines will also be circulated in the `gnuastro-commits` mailing list.

• Like all other Gnuastro’s text files, the lines in the commit body should not be longer than 75 characters, see Section 12.3 [Coding conventions], page 673. This is to ensure that on standard terminal emulators (with 80 character width), the `git log` output can be cleanly displayed (note that the commit message is indented in the output of `git log`). If you use Emacs, Gnuastro’s `.dir-locals.el` file will ensure that your commits satisfy this condition (using `M-q`).

• When the commit is related to a task or a bug, please include the respective ID (in the format of `bug/task #ID`, note the space) in the commit message (from Section 12.10 [Gnuastro project webpage], page 691) for interested people to be able to followup the discussion that took place there. If the commit fixes a bug or finishes a task, the recommended way is to add a line after the body with ‘This fixes bug #ID.’, or ‘This finishes task #ID.’. Don’t assume that the reader has internet access to check the

---

29 http://catb.org/esr/writings/unix-koans/prodigy.html
bug’s full description when reading the commit message, so give a short introduction too.

Below you can see a good commit message example (don’t forget to read it, it has tips for you). After reading this, please run git log on the master branch and read some of the recent commits for more realistic examples.

The first line should be the title of the commit

An empty line is necessary after the title so Git doesn't confuse lines. This top paragraph of the body of the commit usually describes the reason this commit was done. Therefore it usually starts with "Until now ...". It is very useful to explain the reason behind the change, things that aren't immediately obvious when looking into the code. You don't need to list the names of the files, or what lines have been changed, don't forget that the code changes are fully stored within Git :-).

In the second paragraph (or any later paragraph!) of the body, we describe the solution and why (not "how"!) the particular solution was implemented. So we usually start this part of the commit body with "With this commit ...". Again, you don't need to go into the details that can be seen from the 'git diff' command (like the file names that have been changed or the code that has been implemented). The important thing here is the things that aren't immediately obvious from looking into the code.

You can continue the explanation and it is encouraged to be very explicit about the "human factor" of the change as much as possible, not technical details.

12.12.3 Production workflow

Fortunately ‘Pro Git’ has done a wonderful job in explaining the different workflows in Chapter 5\(^{30}\) and in particular the “Integration-Manager Workflow” explained there. The implementation of this workflow is nicely explained in Section 5.2\(^{31}\) under “Forked-Public-Project”. We have also prepared a short tutorial in Section 12.12.4 [Forking tutorial], page 697. Anything on the master branch should always be tested and ready to be built and used. As described in ‘Pro Git’, there are two methods for you to contribute to Gnuastro in the Integration-Manager Workflow:

1. You can send commit patches by email as fully explained in ‘Pro Git’. This is good for your first few contributions. Just note that raw patches (containing only the diff) do not have any meta-data (author name, date, etc). Therefore they will not allow us to fully acknowledge your contributions as an author in Gnuastro: in the AUTHORS file and at the start of the PDF book. These author lists are created automatically from the version controlled source.


To receive full acknowledgment when submitting a patch, is thus advised to use Git’s `format-patch` tool. See Pro Git’s Public project over email (https://git-scm.com/book/en/v2/Distributed-Git-Contributing-to-a-Project#Public-Project-over-Email) section for a nice explanation. If you would like to get more heavily involved in Gnuastro’s development, then you can try the next solution.

2. You can have your own forked copy of Gnuastro on any hosting site you like (Codeberg, Gitlab, GitHub, BitBucket, etc) and inform us when your changes are ready so we merge them in Gnuastro. This is more suited for people who commonly contribute to the code (see Section 12.12.4 [Forking tutorial], page 697).

In both cases, your commits (with your name and information) will be preserved and your contributions will thus be fully recorded in the history of Gnuastro and in the `AUTHORS` file and this book (second page in the PDF format) once they have been incorporated into the official repository. Needless to say that in such cases, be sure to follow the bug or task trackers (or subscribe to the `gnuastro-devel` mailing list) and contact us before hand so you don’t do something that someone else is already working on. In that case, you can get in touch with them and help the job go on faster, see Section 12.10 [Gnuastro project webpage], page 691. This workflow is currently mostly borrowed from the general recommendations of Git\(^\text{32}\) and GitHub. But since Gnuastro is currently under heavy development, these might change and evolve to better suit our needs.

### 12.12.4 Forking tutorial

This is a tutorial on the second suggested method (commonly known as forking) that you can submit your modifications in Gnuastro (see Section 12.12.3 [Production workflow], page 696).

To start, please create an empty repository on your hosting service web page (we recommend Codeberg since it is fully free software\(^\text{33}\)). If this is your first hosted repository on the web page, you also have to upload your public SSH key\(^\text{34}\) for the `git push` command below to work. Here we’ll assume you use the name `janedoe` to refer to yourself everywhere and that you choose `gnuastro` as the name of your Gnuastro fork. Any online hosting service will give you an address (similar to the ‘`git@codeberg.org:`...’ below) of the empty repository you have created using their web page, use that address in the third line below.

```bash
$ git clone git://git.sv.gnu.org/gnuastro.git
$ cd gnuastro
$ git remote add janedoe git@codeberg.org:janedoe/gnuastro.git
$ git push janedoe master
```

The full Gnuastro history is now pushed onto your hosting service and the `janedoe` remote is now also following your `master` branch. If you run `git remote show REMOTENAME` for the `origin` and `janedoe` remotes, you will see their difference: the first has pull access and the second doesn’t. This nicely summarizes the main idea behind this workflow: you

\(^\text{32}\) [https://github.com/git/git/blob/master/Documentation/SubmittingPatches](https://github.com/git/git/blob/master/Documentation/SubmittingPatches)

\(^\text{33}\) See [https://www.gnu.org/software/repo-criteria-evaluation.html](https://www.gnu.org/software/repo-criteria-evaluation.html) for an evaluation of the major existing repositories. Gnuastro uses GNU Savannah (which also has the highest ranking in the evaluation), but for starters, Codeberg may be easier (it is fully free software).

\(^\text{34}\) For example see this explanation provided by Codeberg: [https://docs.codeberg.org/security/ssh-key](https://docs.codeberg.org/security/ssh-key).
push to your remote repository, we pull from it and merge it into **master**, then you finalize it by pulling from the main repository.

To test (compile) your changes during your work, you will need to bootstrap the version controlled source, see Section 3.2.2.1 [Bootstrapping], page 133, for a full description. The cloning process above is only necessary for your first time setup, you don’t need to repeat it. However, please repeat the steps below for each independent issue you intend to work on.

Let’s assume you have found a bug in **lib/statistics.c**’s median calculating function. Before actually doing anything, please announce it (see Section 1.8 [Report a bug], page 11) so everyone knows you are working on it or to find out others aren’t already working on it. With the commands below, you make a branch, checkout to it, correct the bug, check if it is indeed fixed, add it to the staging area, commit it to the new branch and push it to your hosting service. But before all of them, make sure that you are on the **master** branch and that your **master** branch is up to date with the main Gnuastro repository with the first two commands.

```
$ git checkout master
$ git pull
$ git checkout -b bug-median-stats # Choose a descriptive name
$ emacs lib/statistics.c
$ # do your checks here
$ git add lib/statistics.c
$ git commit
$ git push janedoe bug-median-stats
```

Your new branch is now on your hosted repository. Through the respective tacker on Savannah (see Section 12.10 [Gnuastro project webpage], page 691) you can then let the other developers know that your **bug-median-stats** branch is ready. They will pull your work, test it themselves and if it is ready to be merged into the main Gnuastro history, they will merge it into the **master** branch. After that is done, you can simply checkout your local **master** branch and pull all the changes from the main repository. After the pull you can run ‘git log’ as shown below, to see how **bug-median-stats** is merged with master. To finalize, you can push all the changes to your hosted repository and delete the branch:

```
$ git checkout master
$ git pull
$ git log --oneline --graph --decorate --all
$ git push janedoe master
$ git branch -d bug-median-stats # delete local branch
$ git push janedoe --delete bug-median-stats # delete remote branch
```

Just as a reminder, always keep your work on each issue in a separate local and remote branch so work can progress on them independently. After you make your announcement, other people might contribute to the branch before merging it in to **master**, so this is very important. As a final reminder: before starting each issue branch from **master**, be sure to run **git pull** in **master** as shown above. This will enable you to start your branch (work) from the most recent commit and thus simplify the final merging of your work.
Appendix A  Gnuastro programs list

GNU Astronomy Utilities 0.17.44-ad2d9, contains the following programs. They are sorted in alphabetical order and a short description is provided for each program. The description starts with the executable names in *this* font followed by a pointer to the respective section in parenthesis. Throughout this book, they are ordered based on their context, please see the top-level contents for contextual ordering (based on what they do).

**Arithmetic**

(*astarithmetic*, see Section 6.2 [Arithmetic], page 272) For arithmetic operations on multiple (theoretically unlimited) number of datasets (images). It has a large and growing set of arithmetic, mathematical, and even statistical operators (for example $+$, $-$, $\ast$, $/$, $\sqrt{}$, $\log$, $\min$, $\text{average}$, $\text{median}$).

**BuildProgram**

(*astbuildprog*, see Section 11.2 [BuildProgram], page 521) Compile, link and run programs that depend on the Gnuastro library (see Section 11.3 [Gnuastro library], page 525). This program will automatically link with the libraries that Gnuastro depends on, so there is no need to explicitly mention them every time you are compiling a Gnuastro library dependent program.

**ConvertType**

(*astconvertt*, see Section 5.2 [ConvertType], page 211) Convert astronomical data files (FITS or IMH) to and from several other standard image and data formats, for example TXT, JPEG, EPS or PDF.

**Convolve**

(*astconvolve*, see Section 6.3 [Convolve], page 306) Convolve (blur or smooth) data with a given kernel in spatial and frequency domain on multiple threads. Convolve can also do de-convolution to find the appropriate kernel to PSF-match two images.

**CosmicCalculator**

(*astcosmiccal*, see Section 9.1 [CosmicCalculator], page 469) Do cosmological calculations, for example the luminosity distance, distance modulus, comoving volume and many more.

**Crop**

(*astcrop*, see Section 6.1 [Crop], page 259) Crop region(s) from an image and stitch several images if necessary. Inputs can be in pixel coordinates or world coordinates.

**Fits**

(*astfits*, see Section 5.1 [Fits], page 194) View and manipulate FITS file extensions and header keywords.

**MakeCatalog**

(*astmkcatalog*, see Section 7.4 [MakeCatalog], page 392) Make catalog of labeled image (output of NoiseChisel). The catalogs are highly customizable and adding new calculations/columns is very straightforward.

**MakeNoise**

(*astmknoise*, see Section 8.2 [MakeNoise], page 461) Make (add) noise to an image, with a large set of random number generators and any seed.
Appendix A: Gnuastro programs list

MakeProfiles

(astmkprof, see Section 8.1 [MakeProfiles], page 440) Make mock 2D profiles in an image. The central regions of radial profiles are made with a configurable 2D Monte Carlo integration. It can also build the profiles on an over-sampled image.

Match

(astmatch, see Section 7.5 [Match], page 429) Given two input catalogs, find the rows that match with each other within a given aperture (may be an ellipse).

NoiseChisel

(astnoisechisel, see Section 7.2 [NoiseChisel], page 362) Detect signal in noise. It uses a technique to detect very faint and diffuse, irregularly shaped signal in noise (galaxies in the sky), using thresholds that are below the Sky value, see arXiv:1505.01664 (http://arxiv.org/abs/1505.01664).

Query

(astquery, see Section 5.4 [Query], page 249) High-level interface to query pre-defined remote, or external databases, and directly download the required sub-tables on the command-line.

Segment

(astsegment, see Section 7.3 [Segment], page 381) Segment detected regions based on the structure of signal and the input dataset’s noise properties.

Statistics

(aststatistics, see Section 7.1 [Statistics], page 337) Statistical calculations on the input dataset (column in a table, image or datacube).

Table

(asttable, Section 5.3 [Table], page 230) Convert FITS binary and ASCII tables into other such tables, print them on the command-line, save them in a plain text file, or get the FITS table information.

Warp

(astwarp, see Section 6.4 [Warp], page 328) Warp image to new pixel grid. Any projective transformation or Homography can be applied to the input images.

The programs listed above are designed to be highly modular and generic. Hence, they are naturally for lower-level operations. In Gnuastro, higher-level operations (combining multiple programs, or running a program in a special way), are done with installed Bash scripts (all prefixed with astscript-). They can be run just like a program and behave very similarly (with minor differences, see Chapter 10 [Installed scripts], page 482).

astscript-ds9-region

(See Section 10.3 [SAO DS9 region files from table], page 492) Given a table (either as a file or from standard input), create an SAO DS9 region file from the requested positional columns (WCS or image coordinates).

astscript-fits-view

(see Section 10.4 [Viewing FITS file contents with DS9 or TOPCAT], page 494) Given any number of FITS files, this script will either open SAO DS9 (for images or cubes) or TOPCAT (for tables) to view them in a graphic user interface (GUI).

astscript-radial-profile

(See Section 10.2 [Generate radial profile], page 486) Calculate the radial profile of an object within an image. The object can be at any location in the image, using various measures (median, sigma-clipped mean and etc), and the radial distance can also be measured on any general ellipse.
Appendix A: Gnuastro programs list

*astscript-sort-by-night*
(See Section 10.1 [Sort FITS files by night], page 483) Given a list of FITS files, and a HDU and keyword name (for a date), this script separates the files in the same night (possibly over two calendar days).

*astscript-psf-*
The following scripts are used to estimate the extended PSF estimation and subtraction as described in the tutorial Section 2.4 [Building the extended PSF], page 97:

*astscript-psf-select-stars*
(see Section 10.5.2 [Invoking astscript-psf-select-stars], page 500) Find all the stars within an image that are suitable for constructing an extended PSF. If the image has WCS, this script can automatically query Gaia to find the good stars.

*astscript-psf-stamp*
(see Section 10.5.3 [Invoking astscript-psf-stamp], page 503) build a crop (stamp) of a certain width around a star at a certain coordinate in a larger image. This script will do sub-pixel re-positioning to make sure the star is centered and can optionally mask all other background sources).

*astscript-psf-scale-factor*
(see Section 10.5.5 [Invoking astscript-psf-scale-factor], page 508) Given a PSF model, and the central coordinates of a star in an image, find the scale factor that has to be multiplied by the PSF to scale it to that star.

*astscript-psf-unite*
(see Section 10.5.4 [Invoking astscript-psf-unite], page 506) Unite the various components of a PSF into one. Because of saturation and non-linearity, to get a good estimate of the extended PSF, its necessary to construct various parts from different magnitude ranges.

*astscript-psf-subtract*
(see Section 10.5.6 [Invoking astscript-psf-subtract], page 510) Given the model of a PSF and the central coordinates of a star in the image, do sub-pixel re-positioning of the PSF, scale it to the star and subtract it from the image.
Appendix B  Other useful software

In this appendix the installation of programs and libraries that are not direct Gnuastro dependencies are discussed. However they can be useful for working with Gnuastro.

B.1 SAO DS9

SAO DS9 (http://ds9.si.edu) is not a requirement of Gnuastro, it is a FITS image viewer. It is therefore a useful tool to visually inspect the images/cubes of your Gnuastro inputs or outputs (for tables, see Section B.2 [TOPCAT], page 703). In Gnuastro we have an installed script to run DS9 or TOPCAT on any number of FITS files (depending on it being an image or table), see Section 10.4 [Viewing FITS file contents with DS9 or TOPCAT], page 494, (which also includes a .desktop file for GUI integration). After installing DS9, you can easily use that script to open any FITS file (table, image or cube).

Like the other packages, it might already be available in your distribution’s repositories. It is also already pre-compiled in the download section of its web page. Once you download it you can unpack and install (move it to a system recognized directory) with the following commands (x.x.x is the version number):

\[
\begin{align*}
$\text{tar}\ xf\ \text{ds9.linux64.x.x.x.tar.gz} \\
$\text{sudo\ mv\ ds9\ /usr/local/bin}
\end{align*}
\]

To test the build (as long as /usr/local/bin is in your PATH, see Section 3.3.1.2 [Installation directory], page 139), you can simply run the command below.

\[
\begin{align*}
$\text{ds9}
\end{align*}
\]

**Install without root permissions:** If you don’t have root permissions, you can simply replace /usr/local/bin in the command above with $HOME/.local/bin. If this directory isn’t in your PATH, you can simply add it with the command below (in your startup file, e.g., ~/.bashrc). For more on PATH and the startup files, see Section 3.3.1.2 [Installation directory], page 139.

\[
\begin{align*}
\text{export\ PATH=}$\text{HOME/.local/bin:PATH}$
\end{align*}
\]

Below you can see a list of known issues in some operating systems that we have found so far. You should be able to identify any potential error when running DS9 from the command-line like above.

- There might be a complaint about the Xss library, which you can find in your distribution package management system.
- You might also get an XPA related error. In this case, you have to add the following line to your ~/.bashrc and ~/.profile file (you will have to log out and back in again for the latter):

\[
\begin{align*}
\text{export\ XPA\_METHOD=local}
\end{align*}
\]
- Your system may not have the SSL library in its standard library path, in this case, put this command in your startup file (for example ~/.bashrc):

\[
\begin{align*}
\text{export\ LD\_LIBRARY\_PATH=}$\text{LD\_LIBRARY\_PATH:/usr/local/ssl/lib}$
\end{align*}
\]
B.2 TOPCAT

TOPCAT (http://www.star.bris.ac.uk/~mbt/topcat) is not a requirement of Gnuastro, it is a table viewer and plotter (in many input formats, including FITS, VOTable, and others). TOPCAT is therefore a useful tool to visually inspect the tables of your Gnuastro inputs or outputs (for images, see Section B.1 [SAO DS9], page 702). In Gnuastro we have an installed script to run DS9 or TOPCAT on any number of FITS files (depending on it being an image or table), see Section 10.4 [Viewing FITS file contents with DS9 or TOPCAT], page 494, (which also includes a .desktop file for GUI integration). After installing DS9, you can easily use that script to open any FITS file (table, image or cube).

TOPCAT is a very large package with many capabilities to visualize tables (as plots). It also has an extensive documentation (http://www.star.bris.ac.uk/~mbt/topcat/#docs) that you can read for optimally using it. TOPCAT is written in Java, so it just needs a relatively recent (in the last decade) Java Virtual Machine (JVM) and Java Runtime Environment (JRE). Your operating system already has a relatively recent Java installation in its package manager, and there is a large chance that it is already installed. So before trying to install Java, try running TOPCAT. If it complains about not finding a suitable Java environment, then proceed to search your operating system’s package manager.

To install TOPCAT, you just need to run the following two commands. The first .jar file is the main TOPCAT Java ARchive (JAR). JAR is a compressed package of Java files and definitions that should be run with a special Java command. But to avoid bothering users with details of how to call Java, TOPCAT also provides a simple shell script (the second downloaded file below) that is easier to call and will do all the internal checks and call Java properly.

```bash
$ wget http://www.star.bris.ac.uk/~mbt/topcat/topcat-full.jar
$ wget http://www.star.bris.ac.uk/~mbt/topcat/topcat
$ chmod +x topcat
$ ./topcat # Just for a check to see if everything works!
$ sudo mv topcat-full.jar topcat /usr/local/bin/
```

Once the two TOPCAT files are copied in the system-wide directory, you can easily open tables with a command like below from anywhere in your operating system.

```bash
$ topcat table.fits
```

Install without root permissions: If you don’t have root permissions, you can simply replace `/usr/local/bin` in the command above with `$HOME/.local/bin`. If this directory isn’t in your PATH, you can simply add it with the command below (in your startup file, e.g., ~/.bashrc). For more on PATH and the startup files, see Section 3.3.1.2 [Installation directory], page 139.

```bash
export PATH="$HOME/.local/bin:$PATH"
```

B.3 PGPLOT

PGPLOT is a package for making plots in C. It is not directly needed by Gnuastro, but can be used by WCSLIB, see Section 3.1.1.3 [WCSLIB], page 121. As explained in Section 3.1.1.3 [WCSLIB], page 121, you can install WCSLIB without it too. It is very old (the most recent version was released early 2001!), but remains one of the main packages for plotting directly
Appendix B: Other useful software

in C. WCSLIB uses this package to make plots if you want it to make plots. If you are interested you can also use it for your own purposes.

If you want your plotting codes in between your C program, PGPLOT is currently one of your best options. The recommended alternative to this method is to get the raw data for the plots in text files and input them into any of the various more modern and capable plotting tools separately, for example the Matplotlib library in Python or PGFplots in \LaTeX. This will also significantly help code readability. Let’s get back to PGPLOT for the sake of WCSLIB. Installing it is a little tricky (mainly because it is so old!).

You can download the most recent version from the FTP link in its web page\footnote{http://www.astro.caltech.edu/~tjp/pgplot/}. You can unpack it with the \texttt{tar xf} command. Let’s assume the directory you have unpacked it to is PGPLOT, most probably it is: /home/username/Downloads/pgplot/. open the drivers.list file:

```
$ gedit drivers.list
```

Remove the ! for the following lines and save the file in the end:

```
PSDRIV 1 /PS
PSDRIV 2 /VPS
PSDRIV 3 /CPS
PSDRIV 4 /VCPS
XWDRIV 1 /XWINDOW
XWDRIV 2 /XSERVE
```

Don’t choose GIF or VGIF, there is a problem in their codes.

Open the PGPLOT/sys_linux/g77_gcc.conf file:

```
$ gedit PGPLOT/sys_linux/g77_gcc.conf
```

change the line saying: FCOMPL="g77" to FCOMPL="gfortran", and save it. This is a very important step during the compilation of the code if you are in GNU/Linux. You now have to create a folder in /usr/local, don’t forget to replace PGPLOT with your unpacked address:

```
$ su
# mkdir /usr/local/pgplot
# cd /usr/local/pgplot
# cp PGPLOT/drivers.list ./
```

To make the Makefile, type the following command:

```
# PGPLOT/makemake PGPLOT linux g77_gcc
```

It should finish by saying: Determining object file dependencies. You have done the hard part! The rest is easy: run these three commands in order:

```
# make
# make clean
# make cpg
```

Finally you have to place the position of this directory you just made into the LD_LIBRARY_PATH environment variable and define the environment variable PGPLOT_DIR. To do that, you have to edit your .bashrc file:

```
$ cd ~
```
$ gedit .bashrc

Copy these lines into the text editor and save it:

```bash
PGPLOT_DIR="/usr/local/pgplot/"; export PGPLOT_DIR
LD_LIBRARY_PATH=$LD_LIBRARY_PATH:/usr/local/pgplot/
export LD_LIBRARY_PATH
```

You need to log out and log back in again so these definitions take effect. After you logged back in, you want to see the result of all this labor, right? Tim Pearson has done that for you, create a temporary folder in your home directory and copy all the demonstration files in it:

```bash
$ cd ~
$ mkdir temp
$ cd temp
$ cp /usr/local/pgplot/pgdemo* ./
$ ls
```

You will see a lot of pgdemoXX files, where XX is a number. In order to execute them type the following command and drink your coffee while looking at all the beautiful plots! You are now ready to create your own.

```
$ ./pgdemoXX
```
Appendix C  GNU Free Doc. License

Version 1.3, 3 November 2008

https://fsf.org/

Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

0. PREAMBLE

The purpose of this License is to make a manual, textbook, or other functional and useful document free in the sense of freedom: to assure everyone the effective freedom to copy and redistribute it, with or without modifying it, either commercially or non-commercially. Secondarily, this License preserves for the author and publisher a way to get credit for their work, while not being considered responsible for modifications made by others.

This License is a kind of “copyleft”, which means that derivative works of the document must themselves be free in the same sense. It complements the GNU General Public License, which is a copyleft license designed for free software.

We have designed this License in order to use it for manuals for free software, because free software needs free documentation: a free program should come with manuals providing the same freedoms that the software does. But this License is not limited to software manuals; it can be used for any textual work, regardless of subject matter or whether it is published as a printed book. We recommend this License principally for works whose purpose is instruction or reference.

1. APPLICABILITY AND DEFINITIONS

This License applies to any manual or other work, in any medium, that contains a notice placed by the copyright holder saying it can be distributed under the terms of this License. Such a notice grants a world-wide, royalty-free license, unlimited in duration, to use that work under the conditions stated herein. The “Document”, below, refers to any such manual or work. Any member of the public is a licensee, and is addressed as “you”. You accept the license if you copy, modify or distribute the work in a way requiring permission under copyright law.

A “Modified Version” of the Document means any work containing the Document or a portion of it, either copied verbatim, or with modifications and/or translated into another language.

A “Secondary Section” is a named appendix or a front-matter section of the Document that deals exclusively with the relationship of the publishers or authors of the Document to the Document’s overall subject (or to related matters) and contains nothing that could fall directly within that overall subject. (Thus, if the Document is in part a textbook of mathematics, a Secondary Section may not explain any mathematics.) The relationship could be a matter of historical connection with the subject or with related matters, or of legal, commercial, philosophical, ethical or political position regarding them.

The “Invariant Sections” are certain Secondary Sections whose titles are designated, as being those of Invariant Sections, in the notice that says that the Document is released
under this License. If a section does not fit the above definition of Secondary then it is
not allowed to be designated as Invariant. The Document may contain zero Invariant
Sections. If the Document does not identify any Invariant Sections then there are none.

The “Cover Texts” are certain short passages of text that are listed, as Front-Cover
Texts or Back-Cover Texts, in the notice that says that the Document is released under
this License. A Front-Cover Text may be at most 5 words, and a Back-Cover Text may
be at most 25 words.

A “Transparent” copy of the Document means a machine-readable copy, represented
in a format whose specification is available to the general public, that is suitable for
revising the document straightforwardly with generic text editors or (for images com-
posed of pixels) generic paint programs or (for drawings) some widely available drawing
editor, and that is suitable for input to text formatters or for automatic translation to
a variety of formats suitable for input to text formatters. A copy made in an otherwise
Transparent file format whose markup, or absence of markup, has been arranged to
thwart or discourage subsequent modification by readers is not Transparent. An image
format is not Transparent if used for any substantial amount of text. A copy that is
not “Transparent” is called “Opaque”.

Examples of suitable formats for Transparent copies include plain ASCII without
markup, Texinfo input format, LATEX input format, SGML or XML using a publicly
available DTD, and standard-conforming simple HTML, PostScript or PDF designed
for human modification. Examples of transparent image formats include PNG, XCF
and JPG. Opaque formats include proprietary formats that can be read and edited
only by proprietary word processors, SGML or XML for which the DTD and/or pro-
cessing tools are not generally available, and the machine-generated HTML, PostScript
or PDF produced by some word processors for output purposes only.

The “Title Page” means, for a printed book, the title page itself, plus such following
pages as are needed to hold, legibly, the material this License requires to appear in
the title page. For works in formats which do not have any title page as such, “Title Page”
means the text near the most prominent appearance of the work’s title, preceding the
beginning of the body of the text.

The “publisher” means any person or entity that distributes copies of the Document
to the public.

A section “Entitled XYZ” means a named subunit of the Document whose title either
is precisely XYZ or contains XYZ in parentheses following text that translates XYZ in
another language. (Here XYZ stands for a specific section name mentioned below, such
as “Acknowledgements”, “Dedications”, “Endorsements”, or “History”.) To “Preserve
the Title” of such a section when you modify the Document means that it remains a
section “Entitled XYZ” according to this definition.

The Document may include Warranty Disclaimers next to the notice which states that
this License applies to the Document. These Warranty Disclaimers are considered to
be included by reference in this License, but only as regards disclaiming warranties:
any other implication that these Warranty Disclaimers may have is void and has no
effect on the meaning of this License.

2. VERBATIM COPYING
You may copy and distribute the Document in any medium, either commercially or noncommercially, provided that this License, the copyright notices, and the license notice saying this License applies to the Document are reproduced in all copies, and that you add no other conditions whatsoever to those of this License. You may not use technical measures to obstruct or control the reading or further copying of the copies you make or distribute. However, you may accept compensation in exchange for copies. If you distribute a large enough number of copies you must also follow the conditions in section 3.

You may also lend copies, under the same conditions stated above, and you may publicly display copies.

3. COPYING IN QUANTITY

If you publish printed copies (or copies in media that commonly have printed covers) of the Document, numbering more than 100, and the Document’s license notice requires Cover Texts, you must enclose the copies in covers that carry, clearly and legibly, all these Cover Texts: Front-Cover Texts on the front cover, and Back-Cover Texts on the back cover. Both covers must also clearly and legibly identify you as the publisher of these copies. The front cover must present the full title with all words of the title equally prominent and visible. You may add other material on the covers in addition. Copying with changes limited to the covers, as long as they preserve the title of the Document and satisfy these conditions, can be treated as verbatim copying in other respects.

If the required texts for either cover are too voluminous to fit legibly, you should put the first ones listed (as many as fit reasonably) on the actual cover, and continue the rest onto adjacent pages.

If you publish or distribute Opaque copies of the Document numbering more than 100, you must either include a machine-readable Transparent copy along with each Opaque copy, or state in or with each Opaque copy a computer-network location from which the general network-using public has access to download using public-standard network protocols a complete Transparent copy of the Document, free of added material. If you use the latter option, you must take reasonably prudent steps, when you begin distribution of Opaque copies in quantity, to ensure that this Transparent copy will remain thus accessible at the stated location until at least one year after the last time you distribute an Opaque copy (directly or through your agents or retailers) of that edition to the public.

It is requested, but not required, that you contact the authors of the Document well before redistributing any large number of copies, to give them a chance to provide you with an updated version of the Document.

4. MODIFICATIONS

You may copy and distribute a Modified Version of the Document under the conditions of sections 2 and 3 above, provided that you release the Modified Version under precisely this License, with the Modified Version filling the role of the Document, thus licensing distribution and modification of the Modified Version to whoever possesses a copy of it. In addition, you must do these things in the Modified Version:

A. Use in the Title Page (and on the covers, if any) a title distinct from that of the Document, and from those of previous versions (which should, if there were any,
be listed in the History section of the Document). You may use the same title as a previous version if the original publisher of that version gives permission.

B. List on the Title Page, as authors, one or more persons or entities responsible for authorship of the modifications in the Modified Version, together with at least five of the principal authors of the Document (all of its principal authors, if it has fewer than five), unless they release you from this requirement.

C. State on the Title page the name of the publisher of the Modified Version, as the publisher.

D. Preserve all the copyright notices of the Document.

E. Add an appropriate copyright notice for your modifications adjacent to the other copyright notices.

F. Include, immediately after the copyright notices, a license notice giving the public permission to use the Modified Version under the terms of this License, in the form shown in the Addendum below.

G. Preserve in that license notice the full lists of Invariant Sections and required Cover Texts given in the Document's license notice.

H. Include an unaltered copy of this License.

I. Preserve the section Entitled "History", Preserve its Title, and add to it an item stating at least the title, year, new authors, and publisher of the Modified Version as given on the Title Page. If there is no section Entitled "History" in the Document, create one stating the title, year, authors, and publisher of the Document as given on its Title Page, then add an item describing the Modified Version as stated in the previous sentence.

J. Preserve the network location, if any, given in the Document for public access to a Transparent copy of the Document, and likewise the network locations given in the Document for previous versions it was based on. These may be placed in the "History" section. You may omit a network location for a work that was published at least four years before the Document itself, or if the original publisher of the version it refers to gives permission.

K. For any section Entitled "Acknowledgements" or "Dedications", Preserve the Title of the section, and preserve in the section all the substance and tone of each of the contributor acknowledgements and/or dedications given therein.

L. Preserve all the Invariant Sections of the Document, unaltered in their text and in their titles. Section numbers or the equivalent are not considered part of the section titles.

M. Delete any section Entitled "Endorsements". Such a section may not be included in the Modified Version.

N. Do not retitle any existing section to be Entitled "Endorsements" or to conflict in title with any Invariant Section.

O. Preserve any Warranty Disclaimers.

If the Modified Version includes new front-matter sections or appendices that qualify as Secondary Sections and contain no material copied from the Document, you may at your option designate some or all of these sections as invariant. To do this, add their
titles to the list of Invariant Sections in the Modified Version’s license notice. These
titles must be distinct from any other section titles.

You may add a section Entitled “Endorsements”, provided it contains nothing but
endorsements of your Modified Version by various parties—for example, statements of
peer review or that the text has been approved by an organization as the authoritative
definition of a standard.

You may add a passage of up to five words as a Front-Cover Text, and a passage of up
to 25 words as a Back-Cover Text, to the end of the list of Cover Texts in the Modified
Version. Only one passage of Front-Cover Text and one of Back-Cover Text may be
added by (or through arrangements made by) any one entity. If the Document already
includes a cover text for the same cover, previously added by you or by arrangement
made by the same entity you are acting on behalf of, you may not add another; but
you may replace the old one, on explicit permission from the previous publisher that
added the old one.

The author(s) and publisher(s) of the Document do not by this License give permission
to use their names for publicity for or to assert or imply endorsement of any Modified
Version.

5. COMBINING DOCUMENTS

You may combine the Document with other documents released under this License,
under the terms defined in section 4 above for modified versions, provided that you
include in the combination all of the Invariant Sections of all of the original documents,
unmodified, and list them all as Invariant Sections of your combined work in its license
notice, and that you preserve all their Warranty Disclaimers.

The combined work need only contain one copy of this License, and multiple identical
Invariant Sections may be replaced with a single copy. If there are multiple Invariant
Sections with the same name but different contents, make the title of each such section
unique by adding at the end of it, in parentheses, the name of the original author or
publisher of that section if known, or else a unique number. Make the same adjustment
to the section titles in the list of Invariant Sections in the license notice of the combined
work.

In the combination, you must combine any sections Entitled “History” in the vari-
ous original documents, forming one section Entitled “History”; likewise combine any
sections Entitled “Acknowledgements”, and any sections Entitled “Dedications”. You
must delete all sections Entitled “Endorsements.”

6. COLLECTIONS OF DOCUMENTS

You may make a collection consisting of the Document and other documents released
under this License, and replace the individual copies of this License in the various
documents with a single copy that is included in the collection, provided that you
follow the rules of this License for verbatim copying of each of the documents in all
other respects.

You may extract a single document from such a collection, and distribute it individu-
ally under this License, provided you insert a copy of this License into the extracted
document, and follow this License in all other respects regarding verbatim copying of
that document.
7. AGGREGATION WITH INDEPENDENT WORKS

A compilation of the Document or its derivatives with other separate and independent documents or works, in or on a volume of a storage or distribution medium, is called an “aggregate” if the copyright resulting from the compilation is not used to limit the legal rights of the compilation’s users beyond what the individual works permit. When the Document is included in an aggregate, this License does not apply to the other works in the aggregate which are not themselves derivative works of the Document.

If the Cover Text requirement of section 3 is applicable to these copies of the Document, then if the Document is less than one half of the entire aggregate, the Document’s Cover Texts may be placed on covers that bracket the Document within the aggregate, or the electronic equivalent of covers if the Document is in electronic form. Otherwise they must appear on printed covers that bracket the whole aggregate.

8. TRANSLATION

Translation is considered a kind of modification, so you may distribute translations of the Document under the terms of section 4. Replacing Invariant Sections with translations requires special permission from their copyright holders, but you may include translations of some or all Invariant Sections in addition to the original versions of these Invariant Sections. You may include a translation of this License, and all the license notices in the Document, and any Warranty Disclaimers, provided that you also include the original English version of this License and the original versions of those notices and disclaimers. In case of a disagreement between the translation and the original version of this License or a notice or disclaimer, the original version will prevail.

If a section in the Document is Entitled “Acknowledgements”, “Dedications”, or “History”, the requirement (section 4) to Preserve its Title (section 1) will typically require changing the actual title.

9. TERMINATION

You may not copy, modify, sublicense, or distribute the Document except as expressly provided under this License. Any attempt otherwise to copy, modify, sublicense, or distribute it is void, and will automatically terminate your rights under this License.

However, if you cease all violation of this License, then your license from a particular copyright holder is reinstated (a) provisionally, unless and until the copyright holder explicitly and finally terminates your license, and (b) permanently, if the copyright holder fails to notify you of the violation by some reasonable means prior to 60 days after the cessation.

Moreover, your license from a particular copyright holder is reinstated permanently if the copyright holder notifies you of the violation by some reasonable means, this is the first time you have received notice of violation of this License (for any work) from that copyright holder, and you cure the violation prior to 30 days after your receipt of the notice.

Termination of your rights under this section does not terminate the licenses of parties who have received copies or rights from you under this License. If your rights have been terminated and not permanently reinstated, receipt of a copy of some or all of the same material does not give you any rights to use it.
10. FUTURE REVISIONS OF THIS LICENSE

The Free Software Foundation may publish new, revised versions of the GNU Free Documentation License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns. See https://www.gnu.org/licenses/.

Each version of the License is given a distinguishing version number. If the Document specifies that a particular numbered version of this License “or any later version” applies to it, you have the option of following the terms and conditions either of that specified version or of any later version that has been published (not as a draft) by the Free Software Foundation. If the Document does not specify a version number of this License, you may choose any version ever published (not as a draft) by the Free Software Foundation. If the Document specifies that a proxy can decide which future versions of this License can be used, that proxy’s public statement of acceptance of a version permanently authorizes you to choose that version for the Document.

11. RELICENSING

“Massive Multiauthor Collaboration Site” (or “MMC Site”) means any World Wide Web server that publishes copyrightable works and also provides prominent facilities for anybody to edit those works. A public wiki that anybody can edit is an example of such a server. A “Massive Multiauthor Collaboration” (or “MMC”) contained in the site means any set of copyrightable works thus published on the MMC site.

“CC-BY-SA” means the Creative Commons Attribution-Share Alike 3.0 license published by Creative Commons Corporation, a not-for-profit corporation with a principal place of business in San Francisco, California, as well as future copyleft versions of that license published by that same organization.

“Incorporate” means to publish or republish a Document, in whole or in part, as part of another Document.

An MMC is “eligible for relicensing” if it is licensed under this License, and if all works that were first published under this License somewhere other than this MMC, and subsequently incorporated in whole or in part into the MMC, (1) had no cover texts or invariant sections, and (2) were thus incorporated prior to November 1, 2008.

The operator of an MMC Site may republish an MMC contained in the site under CC-BY-SA on the same site at any time before August 1, 2009, provided the MMC is eligible for relicensing.
ADDENDUM: How to use this License for your documents

To use this License in a document you have written, include a copy of the License in the document and put the following copyright and license notices just after the title page:

Copyright (C) year your name.
Permission is granted to copy, distribute and/or modify this document under the terms of the GNU Free Documentation License, Version 1.3 or any later version published by the Free Software Foundation; with no Invariant Sections, no Front-Cover Texts, and no Back-Cover Texts. A copy of the license is included in the section entitled "GNU Free Documentation License".

If you have Invariant Sections, Front-Cover Texts and Back-Cover Texts, replace the "with...Texts." line with this:

with the Invariant Sections being list their titles, with the Front-Cover Texts being list, and with the Back-Cover Texts being list.

If you have Invariant Sections without Cover Texts, or some other combination of the three, merge those two alternatives to suit the situation.

If your document contains nontrivial examples of program code, we recommend releasing these examples in parallel under your choice of free software license, such as the GNU General Public License, to permit their use in free software.
Appendix D GNU Gen. Pub. License v3

Version 3, 29 June 2007

Copyright © 2007 Free Software Foundation, Inc. https://fsf.org/

Everyone is permitted to copy and distribute verbatim copies of this license document, but changing it is not allowed.

Preamble

The GNU General Public License is a free, copyleft license for software and other kinds of works.

The licenses for most software and other practical works are designed to take away your freedom to share and change the works. By contrast, the GNU General Public License is intended to guarantee your freedom to share and change all versions of a program—to make sure it remains free software for all its users. We, the Free Software Foundation, use the GNU General Public License for most of our software; it applies also to any other work released this way by its authors. You can apply it to your programs, too.

When we speak of free software, we are referring to freedom, not price. Our General Public Licenses are designed to make sure that you have the freedom to distribute copies of free software (and charge for them if you wish), that you receive source code or can get it if you want it, that you can change the software or use pieces of it in new free programs, and that you know you can do these things.

To protect your rights, we need to prevent others from denying you these rights or asking you to surrender the rights. Therefore, you have certain responsibilities if you distribute copies of the software, or if you modify it: responsibilities to respect the freedom of others.

For example, if you distribute copies of such a program, whether gratis or for a fee, you must pass on to the recipients the same freedoms that you received. You must make sure that they, too, receive or can get the source code. And you must show them these terms so they know their rights.

Developers that use the GNU GPL protect your rights with two steps: (1) assert copyright on the software, and (2) offer you this License giving you legal permission to copy, distribute and/or modify it.

For the developers' and authors' protection, the GPL clearly explains that there is no warranty for this free software. For both users' and authors' sake, the GPL requires that modified versions be marked as changed, so that their problems will not be attributed erroneously to authors of previous versions.

Some devices are designed to deny users access to install or run modified versions of the software inside them, although the manufacturer can do so. This is fundamentally incompatible with the aim of protecting users' freedom to change the software. The systematic pattern of such abuse occurs in the area of products for individuals to use, which is precisely where it is most unacceptable. Therefore, we have designed this version of the GPL to prohibit the practice for those products. If such problems arise substantially in other domains, we stand ready to extend this provision to those domains in future versions of the GPL, as needed to protect the freedom of users.
Finally, every program is threatened constantly by software patents. States should not allow patents to restrict development and use of software on general-purpose computers, but in those that do, we wish to avoid the special danger that patents applied to a free program could make it effectively proprietary. To prevent this, the GPL assures that patents cannot be used to render the program non-free.

The precise terms and conditions for copying, distribution and modification follow.

**TERMS AND CONDITIONS**

0. Definitions.

“This License” refers to version 3 of the GNU General Public License.

“Copyright” also means copyright-like laws that apply to other kinds of works, such as semiconductor masks.

“The Program” refers to any copyrightable work licensed under this License. Each licensee is addressed as “you”. “Licensees” and “recipients” may be individuals or organizations.

To “modify” a work means to copy from or adapt all or part of the work in a fashion requiring copyright permission, other than the making of an exact copy. The resulting work is called a “modified version” of the earlier work or a work “based on” the earlier work.

A “covered work” means either the unmodified Program or a work based on the Program.

To “propagate” a work means to do anything with it that, without permission, would make you directly or secondarily liable for infringement under applicable copyright law, except executing it on a computer or modifying a private copy. Propagation includes copying, distribution (with or without modification), making available to the public, and in some countries other activities as well.

To “convey” a work means any kind of propagation that enables other parties to make or receive copies. Mere interaction with a user through a computer network, with no transfer of a copy, is not conveying.

An interactive user interface displays “Appropriate Legal Notices” to the extent that it includes a convenient and prominently visible feature that (1) displays an appropriate copyright notice, and (2) tells the user that there is no warranty for the work (except to the extent that warranties are provided), that licensees may convey the work under this License, and how to view a copy of this License. If the interface presents a list of user commands or options, such as a menu, a prominent item in the list meets this criterion.


The “source code” for a work means the preferred form of the work for making modifications to it. “Object code” means any non-source form of a work.

A “Standard Interface” means an interface that either is an official standard defined by a recognized standards body, or, in the case of interfaces specified for a particular programming language, one that is widely used among developers working in that language.
The “System Libraries” of an executable work include anything, other than the work as a whole, that (a) is included in the normal form of packaging a Major Component, but which is not part of that Major Component, and (b) serves only to enable use of the work with that Major Component, or to implement a Standard Interface for which an implementation is available to the public in source code form. A “Major Component”, in this context, means a major essential component (kernel, window system, and so on) of the specific operating system (if any) on which the executable work runs, or a compiler used to produce the work, or an object code interpreter used to run it.

The “Corresponding Source” for a work in object code form means all the source code needed to generate, install, and (for an executable work) run the object code and to modify the work, including scripts to control those activities. However, it does not include the work’s System Libraries, or general-purpose tools or generally available free programs which are used unmodified in performing those activities but which are not part of the work. For example, Corresponding Source includes interface definition files associated with source files for the work, and the source code for shared libraries and dynamically linked subprograms that the work is specifically designed to require, such as by intimate data communication or control flow between those subprograms and other parts of the work.

The Corresponding Source need not include anything that users can regenerate automatically from other parts of the Corresponding Source.

The Corresponding Source for a work in source code form is that same work.

2. Basic Permissions.

All rights granted under this License are granted for the term of copyright on the Program, and are irrevocable provided the stated conditions are met. This License explicitly affirms your unlimited permission to run the unmodified Program. The output from running a covered work is covered by this License only if the output, given its content, constitutes a covered work. This License acknowledges your rights of fair use or other equivalent, as provided by copyright law.

You may make, run and propagate covered works that you do not convey, without conditions so long as your license otherwise remains in force. You may convey covered works to others for the sole purpose of having them make modifications exclusively for you, or provide you with facilities for running those works, provided that you comply with the terms of this License in conveying all material for which you do not control copyright. Those thus making or running the covered works for you must do so exclusively on your behalf, under your direction and control, on terms that prohibit them from making any copies of your copyrighted material outside their relationship with you.

Conveying under any other circumstances is permitted solely under the conditions stated below. Sublicensing is not allowed; section 10 makes it unnecessary.


No covered work shall be deemed part of an effective technological measure under any applicable law fulfilling obligations under article 11 of the WIPO copyright treaty adopted on 20 December 1996, or similar laws prohibiting or restricting circumvention of such measures.
When you convey a covered work, you waive any legal power to forbid circumvention of technological measures to the extent such circumvention is effected by exercising rights under this License with respect to the covered work, and you disclaim any intention to limit operation or modification of the work as a means of enforcing, against the work’s users, your or third parties’ legal rights to forbid circumvention of technological measures.

You may convey verbatim copies of the Program’s source code as you receive it, in any medium, provided that you conspicuously and appropriately publish on each copy an appropriate copyright notice; keep intact all notices stating that this License and any non-permissive terms added in accord with section 7 apply to the code; keep intact all notices of the absence of any warranty; and give all recipients a copy of this License along with the Program.
You may charge any price or no price for each copy that you convey, and you may offer support or warranty protection for a fee.

5. Conveying Modified Source Versions.
You may convey a work based on the Program, or the modifications to produce it from the Program, in the form of source code under the terms of section 4, provided that you also meet all of these conditions:

a. The work must carry prominent notices stating that you modified it, and giving a relevant date.

b. The work must carry prominent notices stating that it is released under this License and any conditions added under section 7. This requirement modifies the requirement in section 4 to “keep intact all notices”.

c. You must license the entire work, as a whole, under this License to anyone who comes into possession of a copy. This License will therefore apply, along with any applicable section 7 additional terms, to the whole of the work, and all its parts, regardless of how they are packaged. This License gives no permission to license the work in any other way, but it does not invalidate such permission if you have separately received it.

d. If the work has interactive user interfaces, each must display Appropriate Legal Notices; however, if the Program has interactive interfaces that do not display Appropriate Legal Notices, your work need not make them do so.

A compilation of a covered work with other separate and independent works, which are not by their nature extensions of the covered work, and which are not combined with it such as to form a larger program, in or on a volume of a storage or distribution medium, is called an “aggregate” if the compilation and its resulting copyright are not used to limit the access or legal rights of the compilation’s users beyond what the individual works permit. Inclusion of a covered work in an aggregate does not cause this License to apply to the other parts of the aggregate.

6. Conveying Non-Source Forms.
You may convey a covered work in object code form under the terms of sections 4 and 5, provided that you also convey the machine-readable Corresponding Source under the terms of this License, in one of these ways:
a. Convey the object code in, or embodied in, a physical product (including a physical distribution medium), accompanied by the Corresponding Source fixed on a durable physical medium customarily used for software interchange.

b. Convey the object code in, or embodied in, a physical product (including a physical distribution medium), accompanied by a written offer, valid for at least three years and valid for as long as you offer spare parts or customer support for that product model, to give anyone who possesses the object code either (1) a copy of the Corresponding Source for all the software in the product that is covered by this License, on a durable physical medium customarily used for software interchange, for a price no more than your reasonable cost of physically performing this conveying of source, or (2) access to copy the Corresponding Source from a network server at no charge.

c. Convey individual copies of the object code with a copy of the written offer to provide the Corresponding Source. This alternative is allowed only occasionally and noncommercially, and only if you received the object code with such an offer, in accord with subsection 6b.

d. Convey the object code by offering access from a designated place (gratis or for a charge), and offer equivalent access to the Corresponding Source in the same way through the same place at no further charge. You need not require recipients to copy the Corresponding Source along with the object code. If the place to copy the object code is a network server, the Corresponding Source may be on a different server (operated by you or a third party) that supports equivalent copying facilities, provided you maintain clear directions next to the object code saying where to find the Corresponding Source. Regardless of what server hosts the Corresponding Source, you remain obligated to ensure that it is available for as long as needed to satisfy these requirements.

e. Convey the object code using peer-to-peer transmission, provided you inform other peers where the object code and Corresponding Source of the work are being offered to the general public at no charge under subsection 6d.

A separable portion of the object code, whose source code is excluded from the Corresponding Source as a System Library, need not be included in conveying the object code work.

A “User Product” is either (1) a “consumer product”, which means any tangible personal property which is normally used for personal, family, or household purposes, or (2) anything designed or sold for incorporation into a dwelling. In determining whether a product is a consumer product, doubtful cases shall be resolved in favor of coverage. For a particular product received by a particular user, “normally used” refers to a typical or common use of that class of product, regardless of the status of the particular user or of the way in which the particular user actually uses, or expects or is expected to use, the product. A product is a consumer product regardless of whether the product has substantial commercial, industrial or non-consumer uses, unless such uses represent the only significant mode of use of the product.

“Installation Information” for a User Product means any methods, procedures, authorization keys, or other information required to install and execute modified versions of a covered work in that User Product from a modified version of its Corresponding Source.
The information must suffice to ensure that the continued functioning of the modified object code is in no case prevented or interfered with solely because modification has been made.

If you convey an object code work under this section in, or with, or specifically for use in, a User Product, and the conveying occurs as part of a transaction in which the right of possession and use of the User Product is transferred to the recipient in perpetuity or for a fixed term (regardless of how the transaction is characterized), the Corresponding Source conveyed under this section must be accompanied by the Installation Information. But this requirement does not apply if neither you nor any third party retains the ability to install modified object code on the User Product (for example, the work has been installed in ROM).

The requirement to provide Installation Information does not include a requirement to continue to provide support service, warranty, or updates for a work that has been modified or installed by the recipient, or for the User Product in which it has been modified or installed. Access to a network may be denied when the modification itself materially and adversely affects the operation of the network or violates the rules and protocols for communication across the network.

Corresponding Source conveyed, and Installation Information provided, in accord with this section must be in a format that is publicly documented (and with an implementation available to the public in source code form), and must require no special password or key for unpacking, reading or copying.

7. Additional Terms.

“Additional permissions” are terms that supplement the terms of this License by making exceptions from one or more of its conditions. Additional permissions that are applicable to the entire Program shall be treated as though they were included in this License, to the extent that they are valid under applicable law. If additional permissions apply only to part of the Program, that part may be used separately under those permissions, but the entire Program remains governed by this License without regard to the additional permissions.

When you convey a copy of a covered work, you may at your option remove any additional permissions from that copy, or from any part of it. (Additional permissions may be written to require their own removal in certain cases when you modify the work.) You may place additional permissions on material, added by you to a covered work, for which you have or can give appropriate copyright permission.

Notwithstanding any other provision of this License, for material you add to a covered work, you may (if authorized by the copyright holders of that material) supplement the terms of this License with terms:

a. Disclaiming warranty or limiting liability differently from the terms of sections 15 and 16 of this License; or

b. Requiring preservation of specified reasonable legal notices or author attributions in that material or in the Appropriate Legal Notices displayed by works containing it; or

c. Prohibiting misrepresentation of the origin of that material, or requiring that modified versions of such material be marked in reasonable ways as different from the original version; or
d. Limiting the use for publicity purposes of names of licensors or authors of the material; or

e. Declining to grant rights under trademark law for use of some trade names, trademarks, or service marks; or

f. Requiring indemnification of licensors and authors of that material by anyone who conveys the material (or modified versions of it) with contractual assumptions of liability to the recipient, for any liability that these contractual assumptions directly impose on those licensors and authors.

All other non-permissive additional terms are considered “further restrictions” within the meaning of section 10. If the Program as you received it, or any part of it, contains a notice stating that it is governed by this License along with a term that is a further restriction, you may remove that term. If a license document contains a further restriction but permits relicensing or conveying under this License, you may add to a covered work material governed by the terms of that license document, provided that the further restriction does not survive such relicensing or conveying.

If you add terms to a covered work in accord with this section, you must place, in the relevant source files, a statement of the additional terms that apply to those files, or a notice indicating where to find the applicable terms.

Additional terms, permissive or non-permissive, may be stated in the form of a separately written license, or stated as exceptions; the above requirements apply either way.

8. Termination.

You may not propagate or modify a covered work except as expressly provided under this License. Any attempt otherwise to propagate or modify it is void, and will automatically terminate your rights under this License (including any patent licenses granted under the third paragraph of section 11).

However, if you cease all violation of this License, then your license from a particular copyright holder is reinstated (a) provisionally, unless and until the copyright holder explicitly and finally terminates your license, and (b) permanently, if the copyright holder fails to notify you of the violation by some reasonable means prior to 60 days after the cessation.

Moreover, your license from a particular copyright holder is reinstated permanently if the copyright holder notifies you of the violation by some reasonable means, this is the first time you have received notice of violation of this License (for any work) from that copyright holder, and you cure the violation prior to 30 days after your receipt of the notice.

Termination of your rights under this section does not terminate the licenses of parties who have received copies or rights from you under this License. If your rights have been terminated and not permanently reinstated, you do not qualify to receive new licenses for the same material under section 10.


You are not required to accept this License in order to receive or run a copy of the Program. Ancillary propagation of a covered work occurring solely as a consequence of using peer-to-peer transmission to receive a copy likewise does not require acceptance.
However, nothing other than this License grants you permission to propagate or modify any covered work. These actions infringe copyright if you do not accept this License. Therefore, by modifying or propagating a covered work, you indicate your acceptance of this License to do so.

10. Automatic Licensing of Downstream Recipients.

Each time you convey a covered work, the recipient automatically receives a license from the original licensors, to run, modify and propagate that work, subject to this License. You are not responsible for enforcing compliance by third parties with this License.

An “entity transaction” is a transaction transferring control of an organization, or substantially all assets of one, or subdividing an organization, or merging organizations. If propagation of a covered work results from an entity transaction, each party to that transaction who receives a copy of the work also receives whatever licenses to the work the party’s predecessor in interest had or could give under the previous paragraph, plus a right to possession of the Corresponding Source of the work from the predecessor in interest, if the predecessor has it or can get it with reasonable efforts.

You may not impose any further restrictions on the exercise of the rights granted or affirmed under this License. For example, you may not impose a license fee, royalty, or other charge for exercise of rights granted under this License, and you may not initiate litigation (including a cross-claim or counterclaim in a lawsuit) alleging that any patent claim is infringed by making, using, selling, offering for sale, or importing the Program or any portion of it.

11. Patents.

A “contributor” is a copyright holder who authorizes use under this License of the Program or a work on which the Program is based. The work thus licensed is called the contributor’s “contributor version”.

A contributor’s “essential patent claims” are all patent claims owned or controlled by the contributor, whether already acquired or hereafter acquired, that would be infringed by some manner, permitted by this License, of making, using, or selling its contributor version, but do not include claims that would be infringed only as a consequence of further modification of the contributor version. For purposes of this definition, “control” includes the right to grant patent sublicenses in a manner consistent with the requirements of this License.

Each contributor grants you a non-exclusive, worldwide, royalty-free patent license under the contributor’s essential patent claims, to make, use, sell, offer for sale, import and otherwise run, modify and propagate the contents of its contributor version.

In the following three paragraphs, a “patent license” is any express agreement or commitment, however denominated, not to enforce a patent (such as an express permission to practice a patent or covenant not to sue for patent infringement). To “grant” such a patent license to a party means to make such an agreement or commitment not to enforce a patent against the party.

If you convey a covered work, knowingly relying on a patent license, and the Corresponding Source of the work is not available for anyone to copy, free of charge and under the terms of this License, through a publicly available network server or other readily accessible means, then you must either (1) cause the Corresponding Source to be so
available, or (2) arrange to deprive yourself of the benefit of the patent license for this particular work, or (3) arrange, in a manner consistent with the requirements of this License, to extend the patent license to downstream recipients. “Knowingly relying” means you have actual knowledge that, but for the patent license, your conveying the covered work in a country, or your recipient’s use of the covered work in a country, would infringe one or more identifiable patents in that country that you have reason to believe are valid.

If, pursuant to or in connection with a single transaction or arrangement, you convey, or propagate by procuring conveyance of, a covered work, and grant a patent license to some of the parties receiving the covered work authorizing them to use, propagate, modify or convey a specific copy of the covered work, then the patent license you grant is automatically extended to all recipients of the covered work and works based on it.

A patent license is “discriminatory” if it does not include within the scope of its coverage, prohibits the exercise of, or is conditioned on the non-exercise of one or more of the rights that are specifically granted under this License. You may not convey a covered work if you are a party to an arrangement with a third party that is in the business of distributing software, under which you make payment to the third party based on the extent of your activity of conveying the work, and under which the third party grants, to any of the parties who would receive the covered work from you, a discriminatory patent license (a) in connection with copies of the covered work conveyed by you (or copies made from those copies), or (b) primarily for and in connection with specific products or compilations that contain the covered work, unless you entered into that arrangement, or that patent license was granted, prior to 28 March 2007.

Nothing in this License shall be construed as excluding or limiting any implied license or other defenses to infringement that may otherwise be available to you under applicable patent law.


If conditions are imposed on you (whether by court order, agreement or otherwise) that contradict the conditions of this License, they do not excuse you from the conditions of this License. If you cannot convey a covered work so as to satisfy simultaneously your obligations under this License and any other pertinent obligations, then as a consequence you may not convey it at all. For example, if you agree to terms that obligate you to collect a royalty for further conveying from those to whom you convey the Program, the only way you could satisfy both those terms and this License would be to refrain entirely from conveying the Program.

13. Use with the GNU Affero General Public License.

Notwithstanding any other provision of this License, you have permission to link or combine any covered work with a work licensed under version 3 of the GNU Affero General Public License into a single combined work, and to convey the resulting work. The terms of this License will continue to apply to the part which is the covered work, but the special requirements of the GNU Affero General Public License, section 13, concerning interaction through a network will apply to the combination as such.

14. Revised Versions of this License.
The Free Software Foundation may publish revised and/or new versions of the GNU General Public License from time to time. Such new versions will be similar in spirit to the present version, but may differ in detail to address new problems or concerns.

Each version is given a distinguishing version number. If the Program specifies that a certain numbered version of the GNU General Public License “or any later version” applies to it, you have the option of following the terms and conditions either of that numbered version or of any later version published by the Free Software Foundation. If the Program does not specify a version number of the GNU General Public License, you may choose any version ever published by the Free Software Foundation.

If the Program specifies that a proxy can decide which future versions of the GNU General Public License can be used, that proxy’s public statement of acceptance of a version permanently authorizes you to choose that version for the Program.

Later license versions may give you additional or different permissions. However, no additional obligations are imposed on any author or copyright holder as a result of your choosing to follow a later version.

15. Disclaimer of Warranty.

THERE IS NO WARRANTY FOR THE PROGRAM, TO THE EXTENT PERMITTED BY APPLICABLE LAW. EXCEPT WHEN OTHERWISE STATED IN WRITING THE COPYRIGHT HOLDERS AND/OR OTHER PARTIES PROVIDE THE PROGRAM “AS IS” WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE ENTIRE RISK AS TO THE QUALITY AND PERFORMANCE OF THE PROGRAM IS WITH YOU. SHOULD THE PROGRAM PROVE DEFECTIVE, YOU ASSUME THE COST OF ALL NECESSARY SERVICING, REPAIR OR CORRECTION.

16. Limitation of Liability.

IN NO EVENT UNLESS REQUIRED BY APPLICABLE LAW OR AGREED TO IN WRITING WILL ANY COPYRIGHT HOLDER, OR ANY OTHER PARTY WHO MODIFIES AND/OR CONVEYS THE PROGRAM AS PERMITTED ABOVE, BE LIABLE TO YOU FOR DAMAGES, INCLUDING ANY GENERAL, SPECIAL, INCIDENTAL OR CONSEQUENTIAL DAMAGES ARISING OUT OF THE USE OR INABILITY TO USE THE PROGRAM (INCLUDING BUT NOT LIMITED TO LOSS OF DATA OR DATA BEING RENDERED INACCURATE OR LOSSES SUSTAINED BY YOU OR THIRD PARTIES OR A FAILURE OF THE PROGRAM TO OPERATE WITH ANY OTHER PROGRAMS), EVEN IF SUCH HOLDER OR OTHER PARTY HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

17. Interpretation of Sections 15 and 16.

If the disclaimer of warranty and limitation of liability provided above cannot be given local legal effect according to their terms, reviewing courts shall apply local law that most closely approximates an absolute waiver of all civil liability in connection with the Program, unless a warranty or assumption of liability accompanies a copy of the Program in return for a fee.
END OF TERMS AND CONDITIONS

How to Apply These Terms to Your New Programs

If you develop a new program, and you want it to be of the greatest possible use to the public, the best way to achieve this is to make it free software which everyone can redistribute and change under these terms.

To do so, attach the following notices to the program. It is safest to attach them to the start of each source file to most effectively state the exclusion of warranty; and each file should have at least the “copyright” line and a pointer to where the full notice is found.

one line to give the program’s name and a brief idea of what it does.
Copyright (C) year name of author

This program is free software: you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation, either version 3 of the License, or (at your option) any later version.

This program is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.

You should have received a copy of the GNU General Public License along with this program. If not, see https://www.gnu.org/licenses/.

Also add information on how to contact you by electronic and paper mail.

If the program does terminal interaction, make it output a short notice like this when it starts in an interactive mode:

program Copyright (C) year name of author
This program comes with ABSOLUTELY NO WARRANTY; for details type ‘show w’.
This is free software, and you are welcome to redistribute it under certain conditions; type ‘show c’ for details.

The hypothetical commands ‘show w’ and ‘show c’ should show the appropriate parts of the General Public License. Of course, your program’s commands might be different; for a GUI interface, you would use an “about box”.

You should also get your employer (if you work as a programmer) or school, if any, to sign a “copyright disclaimer” for the program, if necessary. For more information on this, and how to apply and follow the GNU GPL, see https://www.gnu.org/licenses/.

The GNU General Public License does not permit incorporating your program into proprietary programs. If your program is a subroutine library, you may consider it more useful to permit linking proprietary applications with the library. If this is what you want to do, use the GNU Lesser General Public License instead of this License. But first, please read https://www.gnu.org/licenses/why-not-lgpl.html.
Index: Macros, structures and functions

All Gnuastro library's exported macros start with GAL_, and its exported structures and functions start with gal_. This abbreviation stands for GNU Astronomy Library. The next element in the name is the name of the header which declares or defines them, so to use the gal_array_fset_const function, you have to #include <gnuastro/array.h>. See Section 11.3 [Gnuastro library], page 525, for more. The pthread_barrier constructs are our implementation and are only available on systems that don’t have them, see Section 11.3.2.1 [Implementation of pthread_barrier], page 528.

G

gal_arithmetic............................ 606
gal_arithmetic_operator_string........ 608
gal_arithmetic_set_operator............ 607
gal_array_file_recognized.............. 567
gal_array_name_recognized.............. 567
gal_array_name_recognized_multiext... 567
gal_array_read.......................... 567
gal_array_read_one_ch.................. 568
gal_array_read_one_ch_to_type......... 568
gal_array_read_to_type................ 568
gal_binary_connected_adjacency_list... 643
gal_binary_connected_adjacency_matrix 643
gal_binary_connected_components........ 642
gal_binary_connected_indices(gal_data_t) 643
gal_binary_dilate...................... 642
gal_binary_erode...................... 641
gal_binary_holes_fill.................. 644
gal_binary_holes_label................. 644
gal_binary_open........................ 642
gal_blank_alloc_write............... 540
gal_blank_as_string................... 540
gal_blank_flag........................ 541
gal_blank_flag_apply.................. 541
gal_blank_flag_remove.................. 541
gal_blank_initialize............... 540
gal_blank_initialize_array.......... 540
gal_blank_is......................... 540
gal_blank_number..................... 541
gal_blank_present.................... 541
gal_blank_remove..................... 542
gal_blank_remove_realloc............. 542
gal_blank_remove_rows............... 542
gal_blank_write...................... 540
gal_box_border_from_center........... 619
gal_box_bound_ellipse............... 618
gal_box_bound_ellipse_extent......... 618
gal_box_bound_ellipsoid.............. 619
gal_box_bound_ellipsoid_extent....... 619
gal_box_overlap...................... 619
gal_convolve.spatial................ 649
gal_convolve.spatial_correct_ch_edge 650
gal_cosmology_age.................... 660
gal_cosmology-angular_distance....... 660
gal_cosmology-comoving_volume........ 660
gal_cosmology_critical-density....... 660
gal_cosmology-distance_modulus....... 661
gal_cosmology-luminosity-distance.... 660
gal_cosmology-proper-distance........ 660
gal_cosmology-to_absolute_mag........ 661
gal_cosmology-to_absolute_mag......... 661
gal_cosmology-velocity-from-z........ 661
gal_cosmology-velocity-from-z......... 661
gal_data_alloc........................ 547
gal_data_array_alloc .................. 548
gal_data_array_free................... 549
gal_data_array_ptr_alloc............... 549
gal_data_array_ptr_free............... 549
gal_data_copy.......................... 549
gal_data_copy_string_to_number....... 550
gal_data_copy_to_allocated........... 550
gal_data_copy_to_new_type............ 549
gal_data_copy_to_new_type_free....... 550
gal_data_free.......................... 548
gal_data_free_contents............... 548
gal_data_initialize................... 547
gal_dimension_add_coords............. 551
gal_dimension-collapse-mean.......... 552
gal_dimension-collapse-minmax........ 553
gal_dimension-collapse_number....... 552
gal_dimension-collapse_sum........... 552
gal_dimension-coord_to_index......... 551
gal_dimension-dist-elliptical....... 552
gal_dimension-dist-manhattan......... 552
gal_dimension-dist-radial........... 552
gal_dimension-increment............. 551
gal_dimension-index_to_coord......... 551
gal_dimension-is-different........... 551
gal_dimension-nb_neighbors........... 551
gal_dimension-removes_extra........... 553
gal_dimension-total_size........... 551
gal_ds9-reg-read-polygon............. 661
gal_eps_name_is_eps.................. 592
gal_eps_suffix_is_eps................ 592
gal_eps_to_pt....................... 592
gal_eps_write....................... 592
gal_convolve.spatial................ 574
gal_convolve.spatial_correct_ch_edge 574
gal_convolve.spatial_correct_ch_edge 573
gal_fits-bitpix-to-type............. 575
gal_fits-bitpix-to-type ............. 575

gal_fits-bitpix-to-type............... 575

gal_fits-bitpix-to-type............... 575

gal_fits-bitpix-to-type............... 575

gal_fits-bitpix-to-type............... 575

gal_fits-bitpix-to-type............... 575

gal_fits-bitpix-to-type............... 575

gal_fits-bitpix-to-type............... 575

Index: Macros, structures and functions

...
Index: Macros, structures and functions

gal_type_min . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 534
gal_type_name . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 534
gal_type_out . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 535
gal_type_sizeof . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 534
gal_type_string_to_number. . . . . . . . . . . . . . . . . . 536
gal_type_to_string . . . . . . . . . . . . . . . . . . . . . . . . . . 535
gal_units_au_to_ly . . . . . . . . . . . . . . . . . . . . . . . . . . 657
gal_units_au_to_pc . . . . . . . . . . . . . . . . . . . . . . . . . . 656
gal_units_counts_to_jy . . . . . . . . . . . . . . . . . . . . . 656
gal_units_counts_to_mag . . . . . . . . . . . . . . . . . . . . 656
gal_units_dec_to_degree . . . . . . . . . . . . . . . . . . . . 655
gal_units_degree_to_dec . . . . . . . . . . . . . . . . . . . . 656
gal_units_degree_to_ra . . . . . . . . . . . . . . . . . . . . . 655
gal_units_extract_decimal. . . . . . . . . . . . . . . . . . 655
gal_units_ly_to_au . . . . . . . . . . . . . . . . . . . . . . . . . . 657
gal_units_ly_to_pc . . . . . . . . . . . . . . . . . . . . . . . . . . 656
gal_units_mag_to_counts . . . . . . . . . . . . . . . . . . . . 656
gal_units_pc_to_au . . . . . . . . . . . . . . . . . . . . . . . . . . 656
gal_units_pc_to_ly . . . . . . . . . . . . . . . . . . . . . . . . . . 656
gal_units_ra_to_degree . . . . . . . . . . . . . . . . . . . . . 655
gal_wcs_angular_distance_deg . . . . . . . . . . . . . . 599
gal_wcs_clean_small_errors . . . . . . . . . . . . . . . . 597
gal_wcs_coordsys_convert . . . . . . . . . . . . . . . . . . . 597
gal_wcs_coordsys_from_string . . . . . . . . . . . . . . 597
gal_wcs_coordsys_identify. . . . . . . . . . . . . . . . . . 597
gal_wcs_copy . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 596
gal_wcs_coverage . . . . . . . . . . . . . . . . . . . . . . . . . . . . 599
gal_wcs_create . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 595
gal_wcs_decompose_pc_cdelt . . . . . . . . . . . . . . . . 597
gal_wcs_dimension_name . . . . . . . . . . . . . . . . . . . . . 595
gal_wcs_distortion_convert(struct . . . . . . . . 598
gal_wcs_distortion_from_string . . . . . . . . . . . . 598
gal_wcs_distortion_identify . . . . . . . . . . . . . . . 598
gal_wcs_distortion_to_string . . . . . . . . . . . . . . 598
gal_wcs_img_to_world . . . . . . . . . . . . . . . . . . . . . . . 600
gal_wcs_on_tile . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 596
gal_wcs_pixel_area_arcsec2 . . . . . . . . . . . . . . . . 599
gal_wcs_pixel_scale . . . . . . . . . . . . . . . . . . . . . . . . 599
gal_wcs_read . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 595
gal_wcs_read_fitsptr . . . . . . . . . . . . . . . . . . . . . . . 594
gal_wcs_remove_dimension . . . . . . . . . . . . . . . . . . . 596
gal_wcs_to_cd . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 597
gal_wcs_warp_matrix . . . . . . . . . . . . . . . . . . . . . . . . 596
gal_wcs_world_to_img . . . . . . . . . . . . . . . . . . . . . . . 599
gal_wcs_write . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 596
gal_wcs_write_in_fitsptr . . . . . . . . . . . . . . . . . . . 596
gal_wcs_write_wcsstr . . . . . . . . . . . . . . . . . . . . . . . 595
GAL_ARITHMETIC_FLAG_ENVSEED . . . . . . . . . . . . . . . 600
GAL_ARITHMETIC_FLAG_FREE . . . . . . . . . . . . . . . . . . . 600
GAL_ARITHMETIC_FLAG_INPLACE . . . . . . . . . . . . . . . 600
GAL_ARITHMETIC_FLAG_NUMOK. . . . . . . . . . . . . . . . . . 600
GAL_ARITHMETIC_FLAG_QUIET. . . . . . . . . . . . . . . . . . 600
GAL_ARITHMETIC_FLAGS_BASIC . . . . . . . . . . . . . . . . 600
GAL_ARITHMETIC_OP_ABS . . . . . . . . . . . . . . . . . . . . . . 604
GAL_ARITHMETIC_OP_ACOS . . . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_ACOSH . . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_AND . . . . . . . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_ASIN . . . . . . . . . . . . . . . . . . . . . 603

728

GAL_ARITHMETIC_OP_ASINH . . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_ATAN . . . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_ATAN2 . . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_ATANH . . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_BITAND . . . . . . . . . . . . . . . . . . . 605
GAL_ARITHMETIC_OP_BITLSH . . . . . . . . . . . . . . . . . . . 605
GAL_ARITHMETIC_OP_BITNOT . . . . . . . . . . . . . . . . . . . 605
GAL_ARITHMETIC_OP_BITOR . . . . . . . . . . . . . . . . . . . . 605
GAL_ARITHMETIC_OP_BITRSH . . . . . . . . . . . . . . . . . . . 605
GAL_ARITHMETIC_OP_BITXOR . . . . . . . . . . . . . . . . . . . 605
GAL_ARITHMETIC_OP_BOX_AROUND_ELLIPSE . . . . . 606
GAL_ARITHMETIC_OP_COS . . . . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_COSH . . . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_DEC_TO_DEGREE . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_DEGREE_TO_DEC . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_DEGREE_TO_RA . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_DIVIDE . . . . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_EQ . . . . . . . . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_GE . . . . . . . . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_GT . . . . . . . . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_ISBLANK. . . . . . . . . . . . . . . . . . 602
GAL_ARITHMETIC_OP_LE . . . . . . . . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_LOG . . . . . . . . . . . . . . . . . . . . . . 602
GAL_ARITHMETIC_OP_LOG10 . . . . . . . . . . . . . . . . . . . . 602
GAL_ARITHMETIC_OP_LT . . . . . . . . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_MAKENEW. . . . . . . . . . . . . . . . . . 606
GAL_ARITHMETIC_OP_MAX . . . . . . . . . . . . . . . . . . . . . . 604
GAL_ARITHMETIC_OP_MAXVAL . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_MEAN . . . . . . . . . . . . . . . . . . . . . 604
GAL_ARITHMETIC_OP_MEANVAL. . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_MEDIAN . . . . . . . . . . . . . . . . . . . 604
GAL_ARITHMETIC_OP_MEDIANVAL . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_MIN . . . . . . . . . . . . . . . . . . . . . . 604
GAL_ARITHMETIC_OP_MINUS . . . . . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_MINVAL . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_MKNOISE_POISSON . . . . . . . . 604
GAL_ARITHMETIC_OP_MKNOISE_SIGMA . . . . . . . . . . . 604
GAL_ARITHMETIC_OP_MKNOISE_UNIFORM . . . . . . . . 604
GAL_ARITHMETIC_OP_MODULO . . . . . . . . . . . . . . . . . . . 605
GAL_ARITHMETIC_OP_MULTIPLY . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_NE . . . . . . . . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_NOT . . . . . . . . . . . . . . . . . . . . . . 602
GAL_ARITHMETIC_OP_NUMBER . . . . . . . . . . . . . . . . . . . 604
GAL_ARITHMETIC_OP_NUMBERVAL . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_OR . . . . . . . . . . . . . . . . . . . . . . . 602
GAL_ARITHMETIC_OP_PLUS . . . . . . . . . . . . . . . . . . . . . 601
GAL_ARITHMETIC_OP_POW . . . . . . . . . . . . . . . . . . . . . . 605
GAL_ARITHMETIC_OP_QUANTILE . . . . . . . . . . . . . . . . 604
GAL_ARITHMETIC_OP_RA_TO_DEGREE . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_SIGCLIP_MEAN . . . . . . . . . . . . 604
GAL_ARITHMETIC_OP_SIGCLIP_MEDIAN . . . . . . . . . 604
GAL_ARITHMETIC_OP_SIGCLIP_NUMBER . . . . . . . . . 604
GAL_ARITHMETIC_OP_SIGCLIP_STD . . . . . . . . . . . . . 604
GAL_ARITHMETIC_OP_SIN . . . . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_SINH . . . . . . . . . . . . . . . . . . . . . 603
GAL_ARITHMETIC_OP_SIZE . . . . . . . . . . . . . . . . . . . . . 605
GAL_ARITHMETIC_OP_SQRT . . . . . . . . . . . . . . . . . . . . . 602
GAL_ARITHMETIC_OP_STD . . . . . . . . . . . . . . . . . . . . . . 604


GAL_WCS_LINEAR_MATRIX_INVALID ............ 594
GAL_WCS_LINEAR_MATRIX_PC ............... 594

P

pthread_barrier_destroy .................. 529
pthread_barrier_init ..................... 529
pthread_barrier_t ...................... 529
pthread_barrier_wait ................... 529
pthread_barrierattr_t .................. 528
Index

§
§$HOME .................................................. 171
§$HOME/.local/etc/ .................. 171

--checkconfig .......................... 162
--cite .................................. 162
--config=STR .......................... 138
--disable-guide-message ........... 138
--disable-progname .................. 138
--dontdelete .......................... 158
--enable-check-with-valgrind ...... 137
--enable-debug ...................... 137
--enable-gnulibcheck .............. 138, 149
--enable-guide-message=no ....... 137
--enable-progname .................. 138
--enable-progname=no .............. 138
--enable-reentrant .................. 120
--hdu=STR/INT ......................... 157
--help .................................. 153, 162, 173
--help output customization ...... 173
--ignorecase ........................ 158
--keepinputdir ........................ 158, 191
--lastconfig .......................... 164
--log .................................. 166
--numthreads .......................... 169, 175
--numthread=INT ...................... 166
--onlyversion=STR ................... 165
--output ................................ 169
--output=STR .......................... 158
--prefix ............................... 139
--printparams ........................ 155, 163
--program-prefix .................... 144
--program-suffix ..................... 144
--program-transform-name .......... 144
--quiet ................................ 162
--searchin=STR ....................... 157
--setdirconf ......................... 164, 171
--setuserconf ........................ 164, 171
--stdinttimeout ...................... 157
--tableformat=STR ................... 158
--type=STR ............................. 158
--usage ................................ 153, 162, 172
--version .............................. 162
--wcslinearmatrix=STR .............. 159
--without-libgit2 ................... 138
--without-libjpeg ................... 139
--without-libtiff ................... 139
--without-pgplot ................... 122
--7 .................................. 162
--D .................................. 158
--h STR/INT ............................ 157

-I ............................................. 158
-K ........................................... 158
-mecube (DS9) ......................... 495
-N INT .................................. 166
-o STR .................................. 158
-P ........................................ 163
-q ...................................... 162
-s STR .................................. 157
-S ........................................ 164
-t STR .................................. 158
-T STR .................................. 158
-U ..................................... 164
-V ..................................... 162

./gnuastro/ .......................... 171
./configure ........................... 136, 140
./configure options ............... 137
.bashrc ................................ 174, 465

2
2D histogram ........................... 63, 338, 638
2MASS All-Sky Catalog ............... 253

3
32-bit .................................. 527
3D data-cubes .......................... 288, 426

6
64-bit .................................. 527

A
A4 paper size .......................... 148
A4 print book .......................... 148
AAVSO Photometric All Sky Survey, DR9 ... 253
AB magnitude .......................... 280, 395
AB Magnitude .......................... 656
Abd al-rahman Suﬁ ........................................ 18
Abraham de Moivre .......................... 311
ACS ................................... 329
ACS camera ................................ 403
Additions to Gnuastro .................. 13
Adjacency matrix ........................ 643
Adobe systems .......................... 212
ADQL (Astronomical Data Query Language) ................. 249
ADU .................................. 394, 464
Advanced camera for surveys .......... 329
Advanced Camera for Surveys .......... 330
<table>
<thead>
<tr>
<th>Term</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Camera</td>
<td>331</td>
</tr>
<tr>
<td>CANDELS survey</td>
<td>264, 403</td>
</tr>
<tr>
<td>Caspar Wessel</td>
<td>311</td>
</tr>
<tr>
<td>Catalog, Gaia</td>
<td>252</td>
</tr>
<tr>
<td>Catalog, Vizier</td>
<td>253</td>
</tr>
<tr>
<td>CatWISE 2020 catalog</td>
<td>253</td>
</tr>
<tr>
<td>CCF</td>
<td>525</td>
</tr>
<tr>
<td>CCD</td>
<td>190, 329</td>
</tr>
<tr>
<td>CDELT</td>
<td>34, 159</td>
</tr>
<tr>
<td>CDS, VizieR</td>
<td>253</td>
</tr>
<tr>
<td>Celestial sphere</td>
<td>397</td>
</tr>
<tr>
<td>CentOS</td>
<td>129</td>
</tr>
<tr>
<td>Central management</td>
<td>691</td>
</tr>
<tr>
<td>CFITSIO</td>
<td>120, 205, 526, 572</td>
</tr>
<tr>
<td>CFITSIO version on outputs</td>
<td>191</td>
</tr>
<tr>
<td>Change converted pixel values</td>
<td>228</td>
</tr>
<tr>
<td>Channel</td>
<td>190, 568</td>
</tr>
<tr>
<td>Channel, color</td>
<td>214</td>
</tr>
<tr>
<td>Charge-coupled device</td>
<td>329</td>
</tr>
<tr>
<td>Check</td>
<td>1</td>
</tr>
<tr>
<td>Check center of crop</td>
<td>269</td>
</tr>
<tr>
<td>Checking detection algorithms</td>
<td>440</td>
</tr>
<tr>
<td>Checking tests</td>
<td>148</td>
</tr>
<tr>
<td>Checksum</td>
<td>77</td>
</tr>
<tr>
<td>CHECKSUM: FITS keyword</td>
<td>205</td>
</tr>
<tr>
<td>CII doublet</td>
<td>477, 657</td>
</tr>
<tr>
<td>Citation information</td>
<td>680</td>
</tr>
<tr>
<td>CIV doublet</td>
<td>477</td>
</tr>
<tr>
<td>Claudius Ptolemy</td>
<td>18</td>
</tr>
<tr>
<td>CLI: command-line user interface</td>
<td>9</td>
</tr>
<tr>
<td>CLI: repeating operations</td>
<td>10</td>
</tr>
<tr>
<td>Clump</td>
<td>647</td>
</tr>
<tr>
<td>Clump magnitude limit</td>
<td>398</td>
</tr>
<tr>
<td>CMYK</td>
<td>214</td>
</tr>
<tr>
<td>Coaddition</td>
<td>283</td>
</tr>
<tr>
<td>Color</td>
<td>214</td>
</tr>
<tr>
<td>Color channel</td>
<td>214, 568</td>
</tr>
<tr>
<td>Color-magnitude diagram</td>
<td>63, 341</td>
</tr>
<tr>
<td>Colormap</td>
<td>214</td>
</tr>
<tr>
<td>Colormap, gray-scale</td>
<td>214</td>
</tr>
<tr>
<td>Colormap, HSV</td>
<td>214</td>
</tr>
<tr>
<td>Colormap: SLS</td>
<td>227</td>
</tr>
<tr>
<td>Colormap: SLS-inverse</td>
<td>228</td>
</tr>
<tr>
<td>Colormap: Viridis</td>
<td>227</td>
</tr>
<tr>
<td>Colors, broad-band photometry</td>
<td>29</td>
</tr>
<tr>
<td>Colorspace, gray-scale</td>
<td>227</td>
</tr>
<tr>
<td>Colorspace, HSV</td>
<td>227</td>
</tr>
<tr>
<td>Command-line arguments</td>
<td>153</td>
</tr>
<tr>
<td>Command-line help</td>
<td>172</td>
</tr>
<tr>
<td>Command-line options</td>
<td>153</td>
</tr>
<tr>
<td>Command-line scroll</td>
<td>173</td>
</tr>
<tr>
<td>Command-line searching text</td>
<td>173</td>
</tr>
<tr>
<td>Command-line user interface</td>
<td>9</td>
</tr>
<tr>
<td>Command-line, long outputs</td>
<td>173</td>
</tr>
<tr>
<td>Command-line, viewing full book</td>
<td>174</td>
</tr>
<tr>
<td>Comments</td>
<td>27</td>
</tr>
<tr>
<td>Commutative property</td>
<td>331</td>
</tr>
<tr>
<td>Comoving distance</td>
<td>473</td>
</tr>
<tr>
<td>Compare Moffat and Gaussian</td>
<td>443</td>
</tr>
<tr>
<td>Compare Poisson and Gaussian</td>
<td>462</td>
</tr>
<tr>
<td>Compile</td>
<td>1</td>
</tr>
<tr>
<td>Compiled PostScript</td>
<td>212</td>
</tr>
<tr>
<td>Compiler, C</td>
<td>523</td>
</tr>
<tr>
<td>Compiling from source</td>
<td>128</td>
</tr>
<tr>
<td>Completeness</td>
<td>96, 400</td>
</tr>
<tr>
<td>Completion in the shell</td>
<td>166, 680, 685</td>
</tr>
<tr>
<td>Complex numbers</td>
<td>326</td>
</tr>
<tr>
<td>Compression</td>
<td>381, 391</td>
</tr>
<tr>
<td>Concat convex polygons</td>
<td>268, 620</td>
</tr>
<tr>
<td>Configuration file directories</td>
<td>170</td>
</tr>
<tr>
<td>Configuration file format</td>
<td>169</td>
</tr>
<tr>
<td>Configuration file precedence</td>
<td>170</td>
</tr>
<tr>
<td>Configuration file suffix</td>
<td>169</td>
</tr>
<tr>
<td>Configuration, not finding library</td>
<td>149</td>
</tr>
<tr>
<td>Configure options</td>
<td>136</td>
</tr>
<tr>
<td>Configure options particular to Gnuastro</td>
<td>137</td>
</tr>
<tr>
<td>Configuring</td>
<td>136</td>
</tr>
<tr>
<td>Connected component labeling</td>
<td>381, 642</td>
</tr>
<tr>
<td>Connected components</td>
<td>293</td>
</tr>
<tr>
<td>Connectivity</td>
<td>640</td>
</tr>
<tr>
<td>Contour</td>
<td>360</td>
</tr>
<tr>
<td>Convenient book formats</td>
<td>172</td>
</tr>
<tr>
<td>Convention for program source</td>
<td>677</td>
</tr>
<tr>
<td>Converting data formats</td>
<td>211</td>
</tr>
<tr>
<td>Converting image formats</td>
<td>211</td>
</tr>
<tr>
<td>Convex Hull</td>
<td>621</td>
</tr>
<tr>
<td>Convex polygons</td>
<td>268, 620</td>
</tr>
<tr>
<td>Convolution</td>
<td>306, 307, 348, 442</td>
</tr>
<tr>
<td>Convolution kernel</td>
<td>584</td>
</tr>
<tr>
<td>Cookbook</td>
<td>17</td>
</tr>
<tr>
<td>Coordinate matching</td>
<td>630</td>
</tr>
<tr>
<td>Coordinate scales</td>
<td>34</td>
</tr>
<tr>
<td>Coordinate system: Ecliptic</td>
<td>210, 594</td>
</tr>
<tr>
<td>Coordinate system: Equatorial</td>
<td>210, 594</td>
</tr>
<tr>
<td>Coordinate system: Galactic</td>
<td>210, 594</td>
</tr>
<tr>
<td>Coordinate system: Supergalactic</td>
<td>210, 594</td>
</tr>
<tr>
<td>Coordinate transformation</td>
<td>329</td>
</tr>
<tr>
<td>Coordinates, homogeneous</td>
<td>330</td>
</tr>
<tr>
<td>Coordinates, homogeneou</td>
<td>330</td>
</tr>
<tr>
<td>Copyright</td>
<td>6</td>
</tr>
<tr>
<td>Correlated noise</td>
<td>46, 401, 404</td>
</tr>
<tr>
<td>Correlation</td>
<td>307</td>
</tr>
<tr>
<td>Cosmic ray removal</td>
<td>345</td>
</tr>
<tr>
<td>Cosmic rays</td>
<td>306, 328, 343, 345, 348</td>
</tr>
<tr>
<td>COSMOS survey</td>
<td>259</td>
</tr>
<tr>
<td>Cotes, Roger</td>
<td>311</td>
</tr>
<tr>
<td>Counting error</td>
<td>462</td>
</tr>
<tr>
<td>Counting from zero</td>
<td>156</td>
</tr>
<tr>
<td>Counts</td>
<td>394, 464</td>
</tr>
<tr>
<td>Coverage of image over sky</td>
<td>199</td>
</tr>
<tr>
<td>CPPFLAGS</td>
<td>150, 516, 525</td>
</tr>
<tr>
<td>CPU threads</td>
<td>175</td>
</tr>
<tr>
<td>Index</td>
<td>735</td>
</tr>
<tr>
<td>--------</td>
<td>-----</td>
</tr>
<tr>
<td>CPU threads, number</td>
<td>169</td>
</tr>
<tr>
<td>CPU threads, set number</td>
<td>166</td>
</tr>
<tr>
<td>CPU, using all threads</td>
<td>175</td>
</tr>
<tr>
<td>CRLF line terminator</td>
<td>587</td>
</tr>
<tr>
<td>Crop (<code>astcrop</code>)</td>
<td>259</td>
</tr>
<tr>
<td>Crop a given section of image</td>
<td>262</td>
</tr>
<tr>
<td>Crop part of image</td>
<td>259</td>
</tr>
<tr>
<td>Crop section format</td>
<td>262</td>
</tr>
<tr>
<td>Cubes (3D data)</td>
<td>288, 426</td>
</tr>
<tr>
<td>Cubic spline interpolation</td>
<td>651</td>
</tr>
<tr>
<td>Cumulative Frequency Plot</td>
<td>337</td>
</tr>
<tr>
<td>cURL (downloading tool)</td>
<td>124</td>
</tr>
<tr>
<td>Customize --help output</td>
<td>173</td>
</tr>
<tr>
<td>Customize executable names</td>
<td>144</td>
</tr>
<tr>
<td>Customizing installation</td>
<td>136</td>
</tr>
</tbody>
</table>

**D**

- Dark Energy Survey data release 1 | 253
- Dark level in detectors | 463
- Data | 347
- Data cubes | 288
- Data format conversion | 211
- Data structures | 515
- Data type | 544
- Data’s depth | 402
- Database, Gaia | 252
- Database, VizieR | 253
- Dataset: binary | 640
- DATASUM: FITS keyword | 199, 205, 575
- Date: FITS format | 577
- de Moivre, Abraham | 311, 462
- de Vaucouleurs profile | 444
- Debian | 128
- Debug | 146, 164, 524
- Debugging | 137, 683
- Decimal separator | 37
- Declination | 234, 603, 655, 656
- Default executable search directory | 140
- Default library search directory | 142
- Default option values | 156, 169
- Define section to crop | 262
- Dependencies, Gnuastro | 120
- Depth | 398
- Detached threads | 530
- Detection | 306, 362
- Detections false | 400
- Detector | 331
- `developer-build` | 683, 684
- Development packages | 149
- Diagram, Color-magnitude | 341
- Diffraction limited | 442
- Dilation | 293, 642
- Dilation (image processing) | 81
- Directory, install | 141
- Discrete Fourier transform | 326
- Distance, elliptical/ellipsoidal | 552
- Distance, Manhattan | 552
- Distortion, optical | 328
- Distortion, WCS | 210, 594, 508
- Distribution mode | 346
- Distributions, GNU/Linux | 128
- `dnf` | 129
- Doublet: CIII | 477, 657
- Doublet: CIV | 477
- Doublet: MgII | 477, 657
- Doublet: NII | 476, 657
- Doublet: OII | 477, 657
- Doublet: OIII | 657
- Doublet: OIII (in UV) | 477
- Doublet: OIII (visible) | 476
- Doublet: SiII | 476, 657
- Doublet: SiIII | 477
- Douglas Rushkoff | 3
- Drizzle | 332
- DS9 | 45, 69, 95, 227, 300
- DSS WCS distortion | 594
- Dynamic libraries | 142
- Dynamic linking | 517

**E**

- Ecliptic coordinate system | 210, 594
- Edges, image | 332
- Effective radius | 424, 444
- Efficient use of CPU threads | 176
- Ellipse | 440, 552
- Ellipse area | 424
- Ellipsoid | 441, 552
- Ellipsoidal distance | 552
- Elliptical distance | 441, 552
- Emacs buffers | 674
- Encapsulated PostScript | 212
- Environment | 140
- Environment variable, HOME | 140
- Environment variables | 139, 140, 465
- Epoch time, Unix | 578
- Epoch, Unix time | 200, 234, 483
- EPS | 212, 229, 592, 593
- Equatorial coordinate system | 210, 594
- Erosion | 292, 362, 373, 641
- Erosion (image processing) | 81
- Error in surface brightness | 400
- Error, floating point round-off | 326
- etc | 169
- Euler angles | 441, 619
- Euler, Leonhard | 341
- Exact area resampling | 332
- Executable names | 143
- Exposure time | 396
- eXtreme Deep Field (XDF) survey | 29, 403
F
False detections ........................................ 400
Feature request ........................................ 691
Feature requests ........................................ 13
Fedora .................................................. 129
File flags ................................................ 71
File I/O .................................................. 144
File operations .......................................... 194
File system Hierarchy Standard ......................... 169
file systems, tmpfs ....................................... 145
Filename suffix .......................................... 154
first-in-first-out ....................................... 554, 576, 653
FITS .................................................... 191, 572
FITS or FITS/TXT .......................................... 155
FITS standard ............................................ 120, 333, 544
FITS Tables ............................................... 184
Fitting .................................................... 440
Flag (mask) images .................................... 294
Flags, file ................................................ 71
Flip coordinates ......................................... 329
Floating point error ..................................... 405
Floating point round-off error ......................... 326
FLT ....................................................... 155
Flux ..................................................... 394
Flux to magnitude conversion .......................... 395
Foreground pixels ...................................... 371
FORTRAN ............................................... 544
Fourier spectrum ....................................... 326
Free software ............................................ 6
Free Software Foundation ............................... 693
FSF ...................................................... 693
Full Width at Half Maximum ............................ 442
Function gradient over pixel area ..................... 444
Function groups ......................................... 676
Functions for user interface ......................... 679
FWHM .................................................... 47, 423, 442

G
Gaia catalog ............................................. 252
GAIA Data Release (2 or 3) ............................ 253
Gain ...................................................... 394, 464
Galactic coordinate system ............................ 210, 594
Galaxy profiles ......................................... 444
Galileo, Galilei ......................................... 4
Gaussian ................................................. 365, 383, 409
Gaussian distribution .................................. 347, 442
Gaussian FWHM ........................................ 443
GCC ...................................................... 525
GCC: GNU Compiler Collection ....................... 9, 520, 523, 524, 674
Gedit ..................................................... 27
General file operations ................................ 194
Generalized de Vaucouleur profile .................... 444
Gérard de Vaucouleurs ................................ 444
Git ....................................................... 123, 132, 138, 571, 654
GNOME ................................................. 45, 379
GNOME 3 ................................................ 10
GNU Astronomy Utilities (Gnuastro) ................... 1
GNU Autoconf ........................................... 126, 134, 135, 137, 683
GNU Autoconf Archive ................................ 126, 133
GNU Automake .......................................... 125, 134, 683
GNU Autoreconf ........................................ 146
GNU AWK .............................................. 36, 65, 68, 96, 168, 230, 240, 305, 352, 355, 482, 483, 570
GNU Bash .............................................. 10, 69, 141, 206, 482, 673, 680
GNU Binutils ............................................ 517
GNU build system ..................................... 120, 134, 141, 145, 516, 683
GNU C library .......................................... 9, 125, 134, 138, 145, 174, 408, 518, 648, 674, 678
GNU C Library .......................................... 209
GNU coding standards .................................. 1, 673, 675
GNU Compiler Collection (GCC) ...................... 9, 520, 523, 524, 674
GNU Coreutils ........................................ 175, 246, 672
GNU CPP ............................................... 523
GNU Debugger ......................................... 137
GNU Debugger (GDB) .................................. 147
GNU Emacs ............................................. 11, 27, 174, 675, 676, 677
GNU Free Documentation License .................... 7, 706
GNU free documentation license ..................... 4
GNU General Public License (GPL) .................... 4, 7, 714
GNU Grep ............................................... 173, 203, 478, 681
GNU Gzip ............................................... 381, 391
GNU help2man .......................................... 126
GNU Info ............................................... 30, 174
GNU Libtool ........................................... 123, 126, 134, 150, 517, 519, 521, 523, 683
GNU Make .............................................. 177, 523
GNU Parallel .......................................... 68, 177
GNU Portability Library (Gnulib) ..................... 125, 133, 138, 149, 674
GNU Savannah .......................................... 691
GNU Scientific Library ................................ 120, 465, 630, 650, 652
GNU Sed ............................................... 483
GNU SED ............................................... 482
GNU software documentation ......................... 174
GNU style options ..................................... 154
GNU Tar ............................................... 1
GNU Texinfo ........................................... 6, 126, 134, 149, 150
GNU Wget ............................................... 77
GNU/Linux ............................................ 9
Gnastro coding convention ............................ 673
Gnastro common options ............................... 156
Gnastro major version number ......................... 8
Gnastro program structure convention ............... 677
Gnastro project page .................................. 12
Gnastro test scripts ................................... 684
Gnulib .................................................. 672
Gnulib: GNU Portability Library ..................... 125, 133, 138, 149, 674
GPL ..................................................... 714
GPL Ghostscript ......................................... 122, 124, 150, 213
Gradient over pixel area .............................. 444
Graphic user interface ........................................ 9
Gravitational lensing ........................................ 328
Grayscale ........................................................ 214
Groups of similar functions .................................... 676
GUI: graphic user interface .................................... 9
GUI: repeating operations ...................................... 10
Gzip ................................................................. 1, 132

H
H-alpha .................................................................. 476, 657
H-beta ................................................................. 476, 657
H-delta .................................................................. 476, 657
H-epsilon ............................................................... 477, 657
H-gamma ................................................................ 476, 657
Halted program ..................................................... 11
Hashbang .................................................................. 70
HDD ....................................................................... 144
HDU ....................................................................... 153, 157, 196
Header data unit ..................................................... 153, 157
Header file ............................................................. 674
HEALPix ............................................................... 196, 575
Help ........................................................................ 172
help-gnuastro mailing list ........................................ 175
help-gnuastro@gnu.org .......................................... 175
Hexadecimal encoding ............................................ 226, 592
Hipparchus of Nicaea .............................................. 395
Histogram ................................................................ 337, 346, 637
Histogram, 2D ...................................................... 63, 338, 638
history ..................................................................... 69
HOME/.local/ ....................................................... 140
Homebrew ............................................................. 129
HOME ..................................................................... 140
Homogeneous coordinates ...................................... 330
Homography .......................................................... 330
HSV: Hue Saturation Value ..................................... 214, 227
Hubble Space Telescope (HST), 29, 190, 259, 329, 330
Hue, saturation, value ............................................ 227
Hyper Suprime-Cam ............................................... 190
Hyperbolic functions ............................................. 279

I
IAU, international astronomical union ..................... 194
Identifying outliers .................................................. 349
IFU: Integral Field Unit ......................................... 288, 423, 426
Image ................................................................. 214
Image annotation ................................................... 216
Image blurring ....................................................... 442
Image edges ........................................................... 332
Image format conversion ....................................... 211
Image mosaic ......................................................... 259, 328
Image noise ........................................................... 461
Image tiles ............................................................. 259
Image transformations .......................................... 440
Image's sky coverage ............................................ 199
ImageMagick ........................................................ 127
Imaging surveys ..................................................... 259
Immediate neighbors ............................................. 640
Inconsistent results ................................................ 11
Individual profiles .................................................. 459
info-gnuastro@gnu.org ......................................... 135
INFOPATH ............................................................. 142
Input/Output, file .................................................... 144
Inside-out construction ........................................... 441, 445
Install directory ..................................................... 141
Install with no super-user access ............................ 139
Installation ........................................................... 119
Installation, customizing ........................................ 136
Installed help methods ............................................ 172
Instrumental noise .................................................. 464
Integer overflow ..................................................... 275
Integer, Signed ....................................................... 178
Integral Field Unit ................................................... 423
Integral field unit (IFU) .......................................... 288, 426
Integration over pixel .............................................. 444
Integration to infinity ............................................... 446
Internal default value ............................................. 169
Internally stored option value .................................. 175
Interpolation .......................................................... 332, 650
Interpolation, bi-linear ............................................ 332
Interpolation, bicubic ............................................. 332, 349
Interpolation, nearest-neighbor ............................ 349
Interpolation: Akima spline ................................... 652
Interpolation: monotonic ....................................... 652
Interpolation: Polynomial ....................................... 651
Interpolation: Spline .............................................. 651
Interpolation: Steffen ............................................. 652
Intervals, histogram ............................................... 337
INT ........................................................................ 155
IRAF ....................................................................... 211
ISO C90 .................................................................. 670
Issue ..................................................................... 691
IVOA ..................................................................... 249

J
Jansky (Jy) .............................................................. 656
Jansky's (Jy) ............................................................ 395
Java programming language .................................. 670
Java Virtual Machine (JVM) .................................... 671
Jaynes E. T. ............................................................ 6
JPEG compression quality ....................................... 227, 591
JPEG format .......................................................... 123, 139, 211, 590
JVM: Java virtual machine ..................................... 671

K
k-d tree matching ................................................... 632
K-d tree ................................................................. 625
Ken Thomson ......................................................... 5
Kernel, convolution ................................................ 306, 584
Kernel, Linux ........................................................ 183
Kernighan, Brian .................................................... 670
Modeling stars ........................................ 444
Modifying print book ............................... 148
Modularity ............................................. 513
Moffat beta ............................................ 443
Moffat function ....................................... 443
Moffat FWHM .......................................... 443
Moments ................................................ 404
Monte carlo integration ........................... 445
Mosaicing ............................................... 259, 328
Multi-threaded operation .......................... 624
Multi-threaded programs ........................... 175
Multiextension FITS ................................. 494
Multiple file opening, reentrancy ............. 120
Multiplication, matrix .............................. 331
Multithreaded, Matrix .............................. 329
Multithreaded programming ..................... 527

N
Names of executables .............................. 143
Names, customize .................................. 144
Names, programs .................................. 7
Nanomaggy ........................................... 89, 280
NaN ..................................................... 186, 283, 335, 359, 371, 538, 539, 540, 584,
.................................................................. 623
Narrow-band image .................................. 288
NASA/IPAC Extragalactic Database (NED) .. 252
Naval Observatory Merged
Astrometric Dataset ................................ 253
Navigating source files ............................ 677
Nearest-neighbor interpolation ................. 349
Necessary parameters ............................. 169
NED (NASA/IPAC Extragalactic Database) .. 252
Neighborhood ........................................ 306
Neighbors, immediate .............................. 640
NGC5195 ............................................... 76
OII doublet ............................................ 476, 657
No access to super-user install ................ 139
Noise .................................................... 347, 461
Noise simulation ...................................... 463
Noise, correlated ..................................... 46, 404
Noise, instrumental .................................. 464
Non-commutative operations .................... 331
Non-linearity (CCDs) .............................. 110
Non-volatile memory .............................. 180
Normalizing histogram ............................ 337

mproc .................................................. 175
Number ............................................... 633
Number count ........................................ 57
Number of CPU threads to use .................. 166, 169
Number of threads available .................... 175
Number, version ................................... 7
Numbers, complex .................................. 326
Numbers, psuedo-random ......................... 465
Numbers, random .................................. 464

O
Object magnitude limit ........................... 398
Object oriented programming ................... 670
OII doublet ............................................ 477, 657
OII doublet in UV .................................. 477
OII doublet in visible ............................. 476
On/Off options ..................................... 155
Online help ........................................... 172
Opening ............................................... 92
Opening (Mathematical morphology) ........ 642
Opening multiextension FITS ................... 494
OpenMP .............................................. 528
openSUSE ............................................. 130
Operations on files ............................... 194
Operations, non-commutative .................. 331
Operator, structure de-reference ............. 678
Optical distortion .................................. 328
Optimization ........................................ 524, 683
Optimization flag ................................... 674
Option values ....................................... 155
Optional and mandatory tokens ................. 172
Options ............................................... 305
Options common to all programs ............... 156
Options to programs ............................... 153
Options, abbreviation ............................ 155
Options, GNU style ............................... 154
Options, on/off ..................................... 155
Options, repeated .................................. 155
Options, short (~) and long (--), ............. 154
Order in search directory ....................... 142
Outlier ................................................ 355
Outliers .............................................. 349
Output file names, automatic ................. 191
Output FITS headers ............................. 191
Output, wrong ....................................... 11
Overflow, integer .................................. 275
Oversample .......................................... 24
Oversampling ........................................ 445

P
p ........................................................ 678
Package managers ................................. 128
pacman ............................................... 130
Pan-STARRS Data Release 1 ..................... 253
Paper size, A4 ..................................... 148
Paper size, US letter ............................. 148
Parametric PSFs .................................... 442
Parsecs ............................................. 280, 656

PATH ................................................. 140
PDF .................................................. 212, 229, 592, 593
permutation ......................................... 630
Permutation ......................................... 631
PGFPlots ............................................. 216
PGFPlots (LaTeX package) ....................... 64, 342
PGFPlots in T\LaTeX or \LaTeX .................. 673, 704
PGPLOT .......................................... 703
Index

Phase angle ........................................... 326
photo-electrons ..................................... 345
Photoelectrons ...................................... 331
Photon counting noise .............................. 462
Picture element ..................................... 331
Pipe ................................................. 173
Pixel ............................................... 331
Pixel by pixel making of profiles ............... 441
Pixel mixing ........................................ 306, 331, 332
Pixel scale ......................................... 403
Pixeled graphics .................................... 211
Pixels .............................................. 214
Plain text .......................................... 213
Plot, scatter ....................................... 63
Plot: contour ....................................... 360
Plotting directly in C ............................... 703
Plugin ............................................... 517
PNG standard ....................................... 215
Point pixels ........................................ 332
Point source ........................................ 442
Point spread function .............................. 17, 442
Pointers ............................................. 537
Poisson distribution ................................. 462
Poisson noise ....................................... 298
Poisson, Simeon Denis .............................. 462
Polygons, Concave .................................. 268, 620
Polygons, Convex .................................. 268, 620
Polynomial interpolation ........................... 651
Portable Document format ......................... 212
Portable shell ....................................... 482
Position angle ...................................... 406, 440, 552
POSIX Threads ...................................... 529
POSIX threads ...................................... 528
POSIX threads library ............................... 528
Post-fix notation .................................. 272
Postage stamp images ............................... 259
PostScript .......................................... 212, 229, 592, 593
PostScript vs. PDF ................................ 212
Pre-Processor ....................................... 514
Pre-processor macros ................................ 515
Precedence, configuration files .................. 170
prefix/etc/ ......................................... 171
Primary colors ..................................... 214
printf .............................................. 569
Prior WCS distortion ................................ 594
Probability density function ...................... 337, 346, 462
Profile, profile ..................................... 486
Profiles, galaxies .................................. 444
program-complete.bash ............................ 680
programe-complete.bash ............................ 679
program.c, program.h ............................... 678
programparams ..................................... 678
Program crashing .................................. 11
Program names ..................................... 7
Program structure convention .................... 677
Programming, low level ........................... 672
ProgramName ....................................... 7
Projective transformation ......................... 330
Proper distance .................................... 472
PSF .................................................. 17, 19, 442
PSF image size ..................................... 442
PSF over-sample ................................... 446
PSF width .......................................... 442
PSF, Moffat compared Gaussian .................. 443
Psuedo-random numbers ............................ 465
pthread ............................................. 175
pthread_barrier .................................... 528
Ptolemy, Claudius ................................ 18
Public domain ...................................... 6
Purity .............................................. 96, 400
Puzzle solving scientist ........................... 6
Python Matplotlib ................................ 673, 704
Python programming language .................... 670

Q

qsort .................................................. 623
Quality of compression in JPEG .................... 227
Quantile ............................................ 87, 347, 351, 371, 634
Quantile of the mean ................................ 353
Query .............................................. 249
Qutb al-Din al-Shirazi ............................. 310

R

Radial metric ........................................ 161
Radial profile ...................................... 486
Radial profile on ellipse ........................... 441
Radio astronomy .................................... 251
Radius, effective ................................... 444
RAM ............................................... 180, 531, 643
Random number generation ....................... 91
Random number generator, Seed .................. 415, 445, 452, 465, 466, 468
Random numbers ................................... 464
Random row selection .............................. 247
Raster graphics ..................................... 211
Readout noise ...................................... 464
Red Hat ............................................. 129
Redirection ......................................... 21
Redirection in shell ................................ 70
Redirection of output ............................... 173
Reentrancy, multiple file opening ............... 120
Region file (SAO DS9) .............................. 267, 661
Remembering options .............................. 172
Remote operation .................................. 11
Removing ast from executables .................. 144
Repeated options .................................. 155
Report a bug ....................................... 691
Reproducibility .................................... 247, 413, 671
Reproducible bug reports ......................... 12
Reproducible results ................................ 10
Resampling ......................................... 331
Resource heavy operations ....................... 11
Rest frame wavelength ............................ 480
Rest-frame ......................................... 659
Rest-frame wavelength ............................ 476
<table>
<thead>
<tr>
<th>Index</th>
<th>742</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steradian</td>
<td>397</td>
</tr>
<tr>
<td>Stitch multiple images</td>
<td>259</td>
</tr>
<tr>
<td>Stream editor, SED</td>
<td>144</td>
</tr>
<tr>
<td>Stream: standard input</td>
<td>167</td>
</tr>
<tr>
<td>Stream: standard output</td>
<td>167</td>
</tr>
<tr>
<td>Stroustrup, Bjarne</td>
<td>5, 670</td>
</tr>
<tr>
<td>Structure de-reference operator</td>
<td>678</td>
</tr>
<tr>
<td>Structures</td>
<td>515</td>
</tr>
<tr>
<td>STR</td>
<td>155</td>
</tr>
<tr>
<td>Subaru Telescope</td>
<td>190</td>
</tr>
<tr>
<td>Submit new tracker item</td>
<td>12</td>
</tr>
<tr>
<td>Suffix (filename)</td>
<td>154</td>
</tr>
<tr>
<td>Suffixes, EPS format</td>
<td>212</td>
</tr>
<tr>
<td>Suffixes, JPEG images</td>
<td>211</td>
</tr>
<tr>
<td>Suffixes, PDF format</td>
<td>213</td>
</tr>
<tr>
<td>Suffixes, plain text</td>
<td>213</td>
</tr>
<tr>
<td>Sufi, Abd al-rahman</td>
<td>18</td>
</tr>
<tr>
<td>Sum</td>
<td>633</td>
</tr>
<tr>
<td>Sum for total flux</td>
<td>446</td>
</tr>
<tr>
<td>Supergalactic coordinate system</td>
<td>210, 594</td>
</tr>
<tr>
<td>Superscript, not possible</td>
<td>139</td>
</tr>
<tr>
<td>Support request manager</td>
<td>12</td>
</tr>
<tr>
<td>Surface brightness</td>
<td>94, 397, 402</td>
</tr>
<tr>
<td>Surface brightness error</td>
<td>400</td>
</tr>
<tr>
<td>Surface brightness limit</td>
<td>88, 403, 428</td>
</tr>
<tr>
<td>SUSE Linux Enterprise Server</td>
<td>130</td>
</tr>
<tr>
<td>Symbolic link</td>
<td>144</td>
</tr>
<tr>
<td>System Cache</td>
<td>176</td>
</tr>
<tr>
<td>System wide configuration files</td>
<td>171</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>T</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Table viewer</td>
<td>703</td>
</tr>
<tr>
<td>Tables FITS</td>
<td>184</td>
</tr>
<tr>
<td>Tabs are evil</td>
<td>676</td>
</tr>
<tr>
<td>TAP (Table Access Protocol)</td>
<td>249</td>
</tr>
<tr>
<td>Task tracker</td>
<td>13</td>
</tr>
<tr>
<td>Taxicab metric</td>
<td>161</td>
</tr>
<tr>
<td>Test</td>
<td>1</td>
</tr>
<tr>
<td>Test scripts</td>
<td>684</td>
</tr>
<tr>
<td>Tests, error in converting images</td>
<td>150</td>
</tr>
<tr>
<td>Tests, only one passes</td>
<td>150</td>
</tr>
<tr>
<td>Tests, running</td>
<td>148</td>
</tr>
<tr>
<td>tests/during-dev.sh</td>
<td>683</td>
</tr>
<tr>
<td>\texttt{tex}</td>
<td>150, 212</td>
</tr>
<tr>
<td>TexLive</td>
<td>126</td>
</tr>
<tr>
<td>Thread-safety</td>
<td>624</td>
</tr>
<tr>
<td>Thresholding</td>
<td>640</td>
</tr>
<tr>
<td>TIFF format</td>
<td>124, 139, 212, 590</td>
</tr>
<tr>
<td>TiKZ</td>
<td>217</td>
</tr>
<tr>
<td>Tilde expansion as option values</td>
<td>156</td>
</tr>
<tr>
<td>Time zone</td>
<td>485</td>
</tr>
<tr>
<td>Time, Unix epoch</td>
<td>209, 234, 483</td>
</tr>
<tr>
<td>Timeout</td>
<td>157</td>
</tr>
<tr>
<td>tmpfs file system</td>
<td>145</td>
</tr>
<tr>
<td>Top processing source file</td>
<td>679</td>
</tr>
<tr>
<td>Top root structure</td>
<td>678</td>
</tr>
<tr>
<td>TPD WCS distortion</td>
<td>594</td>
</tr>
<tr>
<td>TPV WCS distortion</td>
<td>210, 594, 508</td>
</tr>
<tr>
<td>Tracker</td>
<td>13, 691</td>
</tr>
<tr>
<td>Trailing space</td>
<td>675</td>
</tr>
<tr>
<td>Transform image</td>
<td>440</td>
</tr>
<tr>
<td>Transformation, affine</td>
<td>330</td>
</tr>
<tr>
<td>Transformation, projective</td>
<td>330</td>
</tr>
<tr>
<td>Trigonometry</td>
<td>279</td>
</tr>
<tr>
<td>Truncation radius</td>
<td>446</td>
</tr>
<tr>
<td>Tutorial</td>
<td>17</td>
</tr>
<tr>
<td>Type</td>
<td>178</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>U</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>U.S. Naval Observatory CCD</td>
<td>253</td>
</tr>
<tr>
<td>Astrograph Catalog</td>
<td>128</td>
</tr>
<tr>
<td>ui.c</td>
<td>679</td>
</tr>
<tr>
<td>ui.h</td>
<td>679</td>
</tr>
<tr>
<td>Uncompress source</td>
<td>1</td>
</tr>
<tr>
<td>Undetected objects</td>
<td>462</td>
</tr>
<tr>
<td>Universal time coordinate (UTC)</td>
<td>485</td>
</tr>
<tr>
<td>Unix epoch time</td>
<td>209, 234, 483, 578</td>
</tr>
<tr>
<td>Unsigned integer</td>
<td>178</td>
</tr>
<tr>
<td>Upper limit magnitude</td>
<td>401</td>
</tr>
<tr>
<td>Upper-limit</td>
<td>55</td>
</tr>
<tr>
<td>US letter paper size</td>
<td>148</td>
</tr>
<tr>
<td>Usage pattern</td>
<td>172</td>
</tr>
<tr>
<td>User interface functions</td>
<td>679</td>
</tr>
<tr>
<td>Using CPU threads</td>
<td>175</td>
</tr>
<tr>
<td>Using multiple CPU cores</td>
<td>175</td>
</tr>
<tr>
<td>Using multiple threads</td>
<td>176</td>
</tr>
<tr>
<td>UTC (Universal time coordinate)</td>
<td>485</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>V</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Valgrind</td>
<td>137, 147</td>
</tr>
<tr>
<td>Values to options</td>
<td>155</td>
</tr>
<tr>
<td>Variance</td>
<td>404</td>
</tr>
<tr>
<td>Vatican library</td>
<td>194</td>
</tr>
<tr>
<td>Vector graphics</td>
<td>212</td>
</tr>
<tr>
<td>Verification, checksum</td>
<td>77</td>
</tr>
<tr>
<td>Version control</td>
<td>11, 132</td>
</tr>
<tr>
<td>Version control systems</td>
<td>123, 138</td>
</tr>
<tr>
<td>Version number</td>
<td>7</td>
</tr>
<tr>
<td>Versioning: Shared library</td>
<td>519</td>
</tr>
<tr>
<td>Viewing trackers</td>
<td>13</td>
</tr>
<tr>
<td>Viridis: Colormap</td>
<td>227</td>
</tr>
<tr>
<td>Virtual console</td>
<td>11</td>
</tr>
<tr>
<td>Visualization</td>
<td>214</td>
</tr>
<tr>
<td>VizieR</td>
<td>253</td>
</tr>
<tr>
<td>void *</td>
<td>543</td>
</tr>
<tr>
<td>Volatile memory</td>
<td>180</td>
</tr>
<tr>
<td>VOTable</td>
<td>252, 703</td>
</tr>
</tbody>
</table>
Index 743

W
Wall-clock time ........................................ 176
Wassel, Caspar ........................................ 311
WAT WCS distortion ................................. 594
Watershed algorithm ................................. 382, 646
Wavelength, rest-frame .............................. 476
WCS ........................................................ 121
WCS distortion ........................................ 210, 594, 598
WCS: World Coordinate System .................. 232
WCSLIB .................................................. 121, 336
Weighted average ...................................... 306
WFC3 ....................................................... 329
White space character ................................ 169
Whole-Sky USNO-B1.0 Catalog .................... 253
Wide Field Camera 3 .................................. 329, 330
William Thomson ..................................... 5
WISE All-Sky data Release ......................... 253
World Coordinate System ............................ 121, 336

World Coordinate System (WCS) .................. 232, 288, 404
Writing configuration files ......................... 169
Wrong output ........................................... 11
Wrong results .......................................... 11

X
XDF survey ................................................ 29, 403

Y
yum ......................................................... 129

Z
Zero point ............................................... 23
Zero point magnitude ................................ 89, 395
zypper, OpenSUSE package manager .......... 130