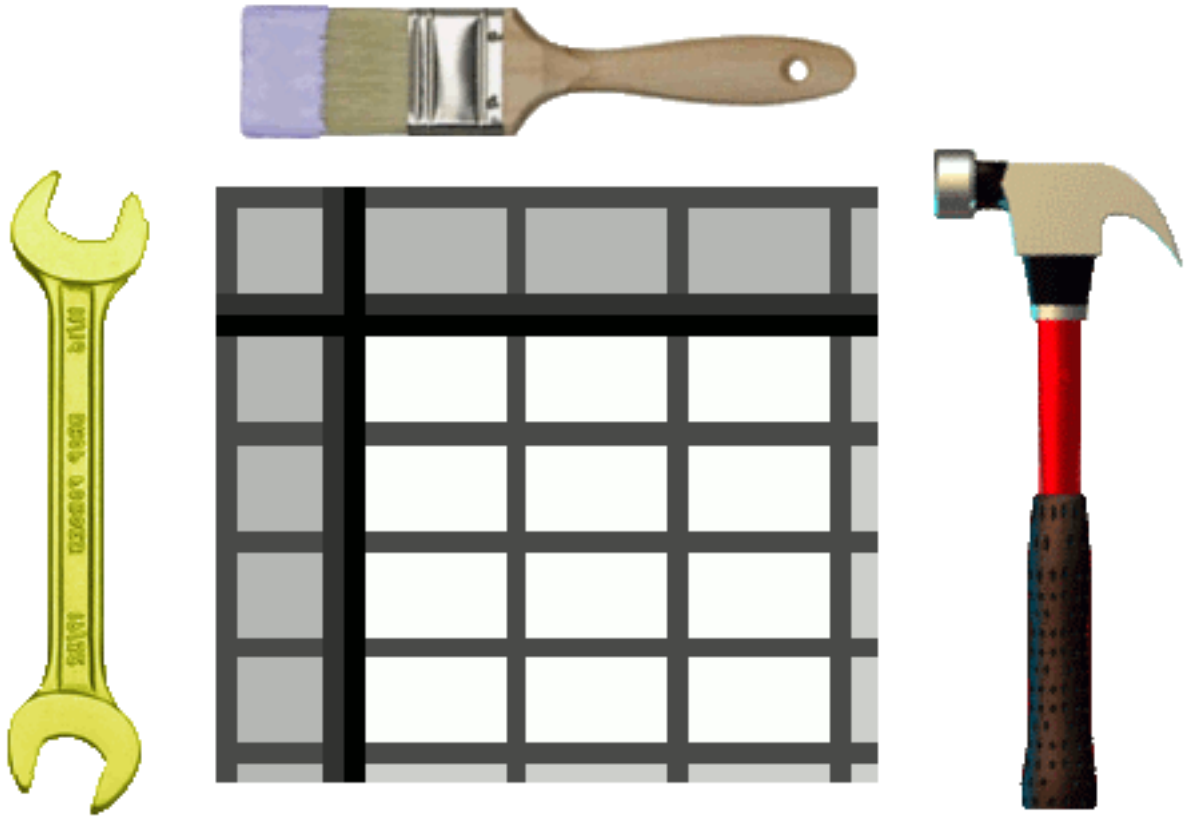
STILTS - Starlink Tables Infrastructure Library Tool Set

Version 3.4-6



Starlink User Note256
Mark Taylor
8 July 2022

Abstract

STILTS is a set of command-line tools for processing tabular data. It has been designed for, but is not restricted to, use on astronomical data such as source catalogues. It contains both generic (format-independent) table processing tools and tools for processing VOTable documents. Facilities offered include crossmatching, format conversion, format validation, column calculation and rearrangement, row selection, sorting, plotting, statistical calculations and metadata display. Calculations on cell data can be performed using a powerful and extensible expression language.

The package is written in pure Java and based on STIL, the Starlink Tables Infrastructure Library. This gives it high portability, support for many data formats (including FITS, VOTable, text-based formats and SQL databases), extensibility and scalability. Where possible the tools are written to accept streamed data so the size of tables which can be processed is not limited by available memory. As well as the tutorial and reference information in this document, detailed on-line help is available from the tools themselves.

The STILTS application is available under the GNU General Public License (GPL) though most parts of the library code may alternatively be used under the GNU Lesser General Public License (LGPL).

Contents

Abstract..... 1
1 Introduction..... 9
2 The `stilts` command..... 11

2.1 Stilts flags ... 11
2.2 Task Names ... 12
2.3 Task Arguments ... 13
2.4 Getting Help ... 14
3 Invocation ... 17
3.1 Class Path ... 18
3.2 Java Flags ... 18
3.3 System Properties ... 19
3.4 JDBC Configuration ... 21
4 JyStilts - STILTS from Python ... 24
4.1 Running JyStilts ... 25
4.2 Table I/O ... 26
4.3 Table objects ... 27
4.4 Table filter commands (`cmd_*`) ... 29
4.5 Table output modes (`mode_*`) ... 30
4.6 Tasks ... 30
4.7 Calculation Functions ... 31
5 Table I/O ... 32
5.1 Table Formats ... 32
5.1.1 Input Formats ... 32
5.1.1.1 `fits` ... 33
5.1.1.2 `colfits` ... 34
5.1.1.3 `votable` ... 34
5.1.1.4 `cdf` ... 35
5.1.1.5 `csv` ... 35
5.1.1.6 `ecsv` ... 37
5.1.1.7 `ascii` ... 38
5.1.1.8 `ipac` ... 39
5.1.1.9 `pds4` ... 40
5.1.1.10 `mrt` ... 41
5.1.1.11 `parquet` ... 42
5.1.1.12 `feather` ... 43
5.1.1.13 `gbin` ... 43
5.1.1.14 `tst` ... 44
5.1.1.15 `wdc` ... 45
5.1.2 Output Formats ... 45
5.1.2.1 `fits` ... 45
5.1.2.2 `colfits` ... 46
5.1.2.3 `votable` ... 47
5.1.2.4 `csv` ... 48
5.1.2.5 `ecsv` ... 48
5.1.2.6 `ascii` ... 49
5.1.2.7 `ipac` ... 50
5.1.2.8 `parquet` ... 51
5.1.2.9 `feather` ... 52
5.1.2.10 `text` ... 52
5.1.2.11 `html` ... 52
5.1.2.12 `latex` ... 53
5.1.2.13 `tst` ... 54
5.1.2.14 `mirage` ... 54
5.2 Input Locations ... 55
5.3 Input Schemes ... 56
5.3.1 `skysim` ... 56
5.3.2 `attractor` ... 57
5.3.3 `jdbc` ... 58

5.3.4 loop 59
5.3.5 test 59
5.3.6 class 60
6 Table Pipelines..... 61
6.1 Processing Filters..... 61
6.1.1 addcol 62
6.1.2 addpixsample 62
6.1.3 addresolve 63
6.1.4 addskycoords 63
6.1.5 assert 64
6.1.6 badval 64
6.1.7 cache 64
6.1.8 check 65
6.1.9 clearparams 65
6.1.10 collapsecols 65
6.1.11 colmeta 66
6.1.12 delcols 66
6.1.13 every 66
6.1.14 explodeall 66
6.1.15 explodecols 67
6.1.16 fixcolnames 67
6.1.17 head 67
6.1.18 healpixmeta 67
6.1.19 keepcols 68
6.1.20 meta 68
6.1.21 progress 69
6.1.22 random 69
6.1.23 randomview 70
6.1.24 repeat 70
6.1.25 replacecol 70
6.1.26 replaceval 70
6.1.27 rowrange 70
6.1.28 select 71
6.1.29 seqview 71
6.1.30 setparam 71
6.1.31 sort 71
6.1.32 sorthead 72
6.1.33 stats 72
6.1.34 tablename 73
6.1.35 tail 73
6.1.36 transpose 73
6.1.37 uniq 74
6.2 Specifying a Single Column..... 74
6.3 Specifying a List of Columns..... 75
6.4 Output Modes..... 75
6.4.1 cgi 76
6.4.2 checksum 76
6.4.3 count 76
6.4.4 discard 76
6.4.5 gui 76
6.4.6 meta 77
6.4.7 out 77
6.4.8 plastic 77
6.4.9 samp 78
6.4.10 stats 78

6.4.11 topcat 79
6.4.12 tosql 79
7 Crossmatching 81
7.1 Match Criteria 81
7.1.1 sky: Sky Matching 82
7.1.2 skyerr: Sky Matching with Per-Object Errors 83
7.1.3 skyellipse: Sky Matching of Elliptical Regions 84
7.1.4 sky3d: Spherical Polar Matching 84
7.1.5 exact: Exact Matching 85
7.1.6 1d, 2d, ...: Isotropic Cartesian Matching 85
7.1.7 2d_anisotropic, ...: Anisotropic Cartesian Matching 86
7.1.8 2d_cuboid, ...: Cuboid Cartesian Matching 86
7.1.9 1d_err, 2d_err, ...: Cartesian Matching with Per-Object Errors 87
7.1.10 2d_ellipse: Cartesian Matching of Elliptical Regions 88
7.1.11 Custom Matchers 88
7.1.12 Matcher Combinations 88
7.2 Multi-Object Matches 89
8 Plotting 91
8.1 Plot Parameters 91
8.1.1 Global Parameters 92
8.1.2 Layer Parameters 92
8.1.3 Animation 93
8.2 Surface Types 94
8.3 Layer Types 95
8.3.1 mark 95
8.3.2 size 98
8.3.3 sizexy 101
8.3.4 xyvector 104
8.3.5 xyerror 107
8.3.6 xyellipse 111
8.3.7 xycorr 114
8.3.8 link2 118
8.3.9 mark2 120
8.3.10 poly4 123
8.3.11 mark4 126
8.3.12 polygon 129
8.3.13 area 132
8.3.14 central 136
8.3.15 lines 139
8.3.16 marks 142
8.3.17 yerrors 144
8.3.18 xyerrors 147
8.3.19 statline 150
8.3.20 statmark 153
8.3.21 arrayquantile 157
8.3.22 line 161
8.3.23 linearfit 164
8.3.24 label 167
8.3.25 arealabel 170
8.3.26 contour 174
8.3.27 grid 178
8.3.28 fill 181
8.3.29 quantile 184
8.3.30 histogram 187
8.3.31 kde 192

8.3.32 knn 196
8.3.33 densogram 201
8.3.34 gaussian 206
8.3.35 function 209
8.3.36 skyvector 211
8.3.37 skyellipse 215
8.3.38 skycorr 219
8.3.39 skydensity 223
8.3.40 healpix 226
8.3.41 skygrid 229
8.3.42 xyzvector 231
8.3.43 xyzerror 234
8.3.44 line3d 238
8.3.45 spheregrid 241
8.3.46 yerror 242
8.3.47 spectrogram 245
8.4 Shading Modes 247
8.4.1 auto 247
8.4.2 flat 249
8.4.3 translucent 250
8.4.4 transparent 251
8.4.5 density 252
8.4.6 aux 255
8.4.7 weighted 257
8.5 Output Modes 258
8.5.1 swing 258
8.5.2 out 259
8.5.3 cgi 259
8.5.4 discard 259
8.5.5 auto 259
8.6 Export Formats 259
9 Old-Style Plotting 261
9.1 Parameter Suffixes 261
10 Algebraic Expression Syntax 264
10.1 Referencing Column Values 264
10.2 Referencing Parameter Values 265
10.3 Special Tokens 266
10.4 Null Values 266
10.5 Operators 267
10.6 Functions 268
10.6.1 Arithmetic 268
10.6.2 Arrays 272
10.6.3 Bits 281
10.6.4 Conversions 282
10.6.5 CoordsDegrees 287
10.6.6 CoordsRadians 290
10.6.7 Coverage 295
10.6.8 Distances 296
10.6.9 Fluxes 299
10.6.10 Formats 302
10.6.11 Gaia 303
10.6.12 KCorrections 312
10.6.13 Lists 315
10.6.14 Maths 317
10.6.15 Randoms 322

10.6.16 Shapes..... 324
10.6.17 Sky..... 325
10.6.18 Strings..... 326
10.6.19 Tilings..... 333
10.6.20 Times..... 338
10.6.21 TrigDegrees..... 344
10.6.22 URLs..... 345
10.6.23 VO..... 349
10.7 Examples..... 350
10.8 Advanced Topics..... 352
10.8.1 Expression evaluation..... 352
10.8.2 Instance Methods..... 352
10.8.3 Adding User-Defined Functions..... 353
11 Server Mode..... 355
11.1 Plot Service..... 355
11.1.1 Usage..... 356
11.1.2 RESTful API..... 356
11.1.3 Caching and Performance..... 359
11.2 Task Service..... 359
11.3 Form Service..... 360
12 Programmatic Invocation..... 362
Appendix A: Commands By Category..... 364
Appendix B: Command Reference..... 366
B.1 `arrayjoin`: Adds table-per-row data as array-valued columns..... 366
B.2 `calc`: Evaluates expressions..... 370
B.3 `cdsskymatch`: Crossmatches table on sky position against VizieR/SIMBAD table..... 371
B.4 `cone`: Executes a Cone Search-like query..... 376
B.5 `coneskymatch`: Crossmatches table on sky position against remote cone service..... 380
B.6 `datalinklint`: Validates DataLink documents..... 387
B.7 `funcs`: Browses functions used by algebraic expression language..... 390
B.8 `pixfoot`: Generates Multi-Order Coverage maps..... 391
B.9 `pixsample`: Samples from a HEALPix pixel data file..... 393
B.10 `plot2plane`: Draws a plane plot..... 397
B.11 `plot2sky`: Draws a sky plot..... 409
B.12 `plot2cube`: Draws a cube plot..... 420
B.13 `plot2sphere`: Draws a sphere plot..... 431
B.14 `plot2time`: Draws a time plot..... 440
B.15 `plot2d`: Old-style 2D Scatter Plot..... 453
B.16 `plot3d`: Old-style 3D Scatter Plot..... 461
B.17 `plothist`: Old-style Histogram..... 470
B.18 `regquery`: Queries the VO registry..... 477
B.19 `server`: Runs an HTTP server to perform STILTS commands..... 479
B.20 `sqlclient`: Executes SQL statements..... 481
B.21 `sqlskymatch`: Crossmatches table on sky position against SQL table..... 482
B.22 `sqlupdate`: Updates values in an SQL table..... 488
B.23 `taplint`: Tests TAP services..... 490
B.24 `tapquery`: Queries a Table Access Protocol server..... 495
B.25 `tapresume`: Resumes a previous query to a Table Access Protocol server..... 500
B.26 `tapskymatch`: Crossmatches table on sky position against TAP table..... 502
B.27 `tcat`: Concatenates multiple similar tables..... 508
B.28 `tcatn`: Concatenates multiple tables..... 512
B.29 `tcopy`: Converts between table formats..... 516
B.30 `tcube`: Calculates N-dimensional histograms..... 518
B.31 `tloop`: Generates a single-column table from a loop variable..... 521
B.32 `tgridmap`: Calculates N-dimensional density maps..... 523

B.33 `tjoin`: Joins multiple tables side-to-side..... 528
B.34 `tmatch1`: Performs a crossmatch internal to a single table..... 531
B.35 `tmatch2`: Crossmatches 2 tables using flexible criteria..... 535
B.36 `tmatchn`: Crossmatches multiple tables using flexible criteria..... 542
B.37 `tmulti`: Writes multiple tables to a single container file..... 547
B.38 `tmultin`: Writes multiple processed tables to single container file..... 550
B.39 `tpipe`: Performs pipeline processing on a table..... 551
B.40 `tskymap`: Calculates sky density maps..... 556
B.41 `tskymatch2`: Crossmatches 2 tables on sky position..... 562
B.42 `votcopy`: Transforms between VOTable encodings..... 566
B.43 `votlint`: Validates VOTable documents..... 570
B.44 `xsdvalidate`: Validates against XML Schema..... 574

Appendix C: Release Notes..... 576
C.1 Acknowledgements..... 576
C.2 Version History..... 576

1 Introduction

STILTS provides a number of command-line applications which can be used for manipulating tabular data. Conceptually it sits between, and uses many of the same classes as, the packages STIL, which is a set of Java APIs providing table-related functionality, and TOPCAT, which is a graphical application providing the user with an interactive platform for exploring one or more tables. This document is mostly self-contained - it covers some of the same ground as the STIL and TOPCAT user documents (SUN/252 and SUN/253 respectively).

Currently, this package consists of commands in the following categories:

Generic table manipulation
: `tcopy`, `tpipe`, `tmulti`, `tmultin`, `tcat`, `tcatn`, `tloop`, `tjoin`, `arrayjoin`, `tgridmap`, and `tcube` (see Section 6).

Crossmatching
: `tmatch1`, `tmatch2`, `tmatchn` and `tskymatch2` (see Section 7).

Plotting
: `plot2plane`, `plot2sky`, `plot2cube`, `plot2sphere` and `plot2time` (also deprecated old-style plot commands `plot2d`, `plot3d` and `plothist`) (see Section 8).

Sky Pixel Operations
: `tskymap`, `pixfoot` and `pixsample`.

VOTable
: `votcopy` and `votlint`.

Virtual Observatory access
: `cdsskymatch`, `cone`, `coneskymatch`, `tapquery`, `tapresume`, `tapskymatch`, `taplint`, `datalinklint` and `regquery`.

SQL databases
: `sqlclient`, `sqlupdate` and `sqlskymatch`.

Miscellaneous
: `calc` (Appendix B.2), `funcs` (Appendix B.7), `server` (Appendix B.19) and `xsdvalidate` (Appendix B.44).

See Appendix A for an expanded version of this list.

There are many ways you might want to use these tools; here are a few possibilities:

In conjunction with TOPCAT
: you can identify a set of processing steps using TOPCAT's interactive graphical facilities, and construct a script using the commands provided here which can perform the same steps on many similar tables without further user intervention.

Format conversion
: If you have a separate table processing engine and you want to be able to output the results in a somewhat different form, for instance converting it from FITS to VOTable or from TABLEDATA-encoded to BINARY-encoded VOTable, or to perform some more scientifically substantial operation such as changing units or coordinate systems, substituting bad values etc, you can pass the results through one of the tools here. Since on the whole operation is streaming, such conversion can easily and efficiently be done on the fly.

Server-side operations
: The tools provided here are suitable for use on servers, either to generate files as part of a web service (perhaps along the lines of the **Format conversion** item above) or as configurable components in a server-based workflow system. The `server` command may help, but is not required, for use in these situations.

Quick look

You might want to examine the metadata, or a few rows, or a statistical summary of a table without having to load the whole thing into TOPCAT or some other table viewer application.

2 The `stilts` command

All the functions available in this package can be used from a single command, which is usually referred to in this document simply as "`stilts`". Depending on how you have installed the package, you may just type "`stilts`", or something like

```
java -jar some/path/stilts.jar
```

or

```
java -classpath topcat-lite.jar uk.ac.starlink.ttools.Stilts
```

or something else - this is covered in detail in Section 3.

In general, the form of a command is

```
stilts <stilts-flags> <task-name> <task-args>
```

The forms of the parts of this command are described in the following subsections, and details of each of the available tasks along with their arguments are listed in the command reference (Appendix B) at the end of this document. Some of the commands are highly configurable and have a variety of parameters to define their operation. In many cases however, it's not complicated to use them. For instance, to convert the data in a FITS table to VOTable format you might write:

```
stilts tcopy cat.fits cat.vot
```

2.1 Stilts flags

Some flags are common to all the tasks in the STILTS package, and these are specified after the `stilts` invocation itself and before the task name. They generally have the same effect regardless of which task is running. These generic flags are as follows:

`-help`
Prints a usage message for the `stilts` command itself and exits. The message contains a listing of all the known tasks.

`-version`
Prints the STILTS version number and exits.

`-verbose`
Causes more verbose information to be written during operation. Specifically, what this does is to boost the logging level by one notch. It may be specified multiple times to increase verbosity further. The flag `+verbose` can be used to do the opposite (reduce the logging level by one notch).

`-allowunused`
Causes unused parameter settings on the command line to be tolerated. Normally, any unused parameters on the command line cause a usage message to be output and the command to fail, on the assumption that if you've supplied a parameter setting that's not doing anything it is probably a mistake and you should be given a chance to correct it. But if this flag is set, you just get a warning through the logging system about any unused parameters, and the command is executed as if they weren't there.

`-prompt`
Most of the STILTS commands have a number of parameters which will assume sensible defaults if you do not give them explicit values on the command line. If you use the `-prompt` flag, then you will be prompted for every parameter you have not explicitly specified to give you an opportunity to enter a value other than the default.

`-bench`
Outputs the elapsed time taken by the task to standard error on successful completion.

`-debug`
Sets up output suitable for debugging. The most visible consequence of this is that if an error occurs then a full stacktrace is output, rather than just a user-friendly report.

`-batch`
Some parameters will prompt you for their values, even if they offer legal defaults. If you use the `-batch` flag, then you won't be prompted at all.

`-memory`
Encourages the command to use java heap memory for caching large amounts of data rather than using temporary disk files. The default is to use memory for small tables, and disk for large ones. This flag is in most cases equivalent to specifying the system property `-Dstartable.storage=memory`.

`-disk`
Encourages the command to use temporary files on disk for caching table data. The default is to use memory for small tables, and disk for large ones. Using this flag may help if you are running out of memory. This flag is in most cases equivalent to specifying the system property `-Dstartable.storage=disk`.

`-memgui`
Displays a graphical window while the command is running which summarises used and available heap memory. May be useful for profiling or understanding resource constraints.

`-checkversion <vers>`
Requires that the version is exactly as given by the string `<vers>`. If it is not, STILTS will exit with an error. This can be useful when executing in certain controlled environments to ensure that the correct version of the application is being picked up.

`-stdout <file>`
Sends all normal output from the run to the given file. By default this goes to the standard output stream. Supplying an empty string or "`-`" for `<file>` will restore this default behaviour.

`-stderr <file>`
Sends all error output from the run to the given file. By default this goes to the standard error stream. Supplying an empty string or "`-`" for `<file>` will restore this default behaviour.

If you are submitting an error report, please include the result of running `stilts -version` and the output of the troublesome command with the `-debug` flag specified.

2.2 Task Names

The `<task-name>` part of the command line is the name of one of the tasks listed in Appendix B - currently the available tasks are:

- `arrayjoin`
- `calc`
- `cdsskymatch`
- `cone`
- `coneskymatch`
- `datalinklint`
- `funcs`
- `pixfoot`
- `pixsample`
- `plot2cube`
- `plot2plane`

- plot2sphere
- plot2sky
- plot2time
- plot2d
- plot3d
- plothist
- regquery
- server
- sqlclient
- sqlskymatch
- sqlupdate
- taplint
- tapquery
- tapresume
- tapskymatch
- tcat
- tcatn
- tcopy
- tcube
- tgridmap
- tjoin
- tloop
- tmatch1
- tmatch2
- tmatchn
- tmulti
- tmultin
- tpipe
- tskymap
- tskymatch2
- votcopy
- votlint
- xsdvalidate

2.3 Task Arguments

The <task-args> part of the command line is a list of parameter assignments, each giving the value of one of the named parameters belonging to the task which is specified in the <task-name> part.

The general form of each parameter assignment is

```
<param-name>=<param-value>
```

If you want to set the parameter to the null value, which is legal for some but not all parameters, use the special string "null", or just leave the value blank ("<param-name>="). In some cases you can optionally leave out the <param-name> part of the assignment (i.e. the parameter is positionally determined); this is indicated in the task's usage description if the parameter is described like [<param-name>=]<param-value> rather than <param-name>=<param-value>. If the <param-value> contains spaces or other special characters, then in most cases, such as from the Unix shell, you will have to quote it somehow. How this is done depends on your platform, but usually surrounding the whole value in single quotes will do the trick.

Tasks may have many parameters, and you don't have to set all of them explicitly on the comand line. For a parameter which you don't set, two things can happen. In many cases, it will default to some sensible value. Sometimes however, you may be prompted for the value to use. In the latter

case, a line like this will be written to the terminal:

```
matcher - Name of matching algorithm [sky]:
```

This is prompting you for the value of the parameter named `matcher`. "Name of matching algorithm" is a short description of what that parameter does. "`sky`" is the default value (if there is no default, no value will appear in square brackets). At this point you can do one of four things:

- Hit return - this will select the default value if there is one. If there is no default, this is equivalent to entering "`null`".
- Enter a value for the parameter explicitly. The special value "`null`" means the null value, which is legal for some, but not all parameters. If the value you enter is not legal, you will see an error message and you will be invited to try again.
- Enter "`help`" or a question mark "`?`". This will output a message giving a detailed description of the parameter and prompt you again.
- Bail out by hitting ctrl-C or whatever is usual on your platform.

Under normal circumstances, most parameters which have a legal default value will default to it if they are not set on the command line, and you will only be prompted for those where there is no default or the program thinks there's a good chance you might not want to use it. You can influence this however using flags to the `stilts` command itself (see Section 2.1). If you supply the `-prompt` flag, then you will be prompted for every parameter you have not explicitly set. If you supply `-batch` on the other hand, you won't be prompted for any parameters (and if you fail to set any without legal default values, the task will fail).

If you want to see the actual values of the parameters for a task as it runs, including prompted values and defaulted ones which you haven't specified explicitly, you can use the `-verbose` flag after the `stilts` command:

```
% stilts -verbose tcopy cat.fits cat.vot ifmt=fits
INFO: tcopy in=cat.fits out=cat.vot ifmt=fits ofmt=(auto)
```

If you make a parameter assignment on the command line for a parameter which is not used by the task in question, STILTS will issue an error message and the task will fail. Note some parameters are only used dependent on the presence or values of other parameters, so even supplying a parameter which is documented in the task's usage can have this effect. This is done on the assumption that if you have supplied a spurious parameter it's probably a mistake and you should be given the opportunity to correct it. But if you want to be free to make these mistakes without the task failing, you can supply the `-allowunused` flag as described in Section 2.1, in which case they will just result in a warning.

Extensive help is available from `stilts` itself about task and its parameters, as described in the next section.

2.4 Getting Help

As well as the command descriptions in this document (especially the reference section Appendix B) you can get help for STILTS usage from the command itself. Typing

```
stilts -help
```

results in this output:

```
Usage:
   stilts [-help] [-version] [-verbose] [-allowunused] [-prompt] [-bench]
          [-debug] [-batch] [-memory] [-disk] [-memgui]
          [-checkversion <vers>] [-stdout <file>] [-stderr <file>]
          <task-name> <task-args>
```

```
   stilts <task-name> help[=<param-name>|*]

   Known tasks:
      arrayjoin
      calc
      cdsskymatch
      cone
      coneskymatch
      datalinklint
      funcs
      pixfoot
      pixsample
      plot2d
      plot3d
      plothist
      regquery
      server
      sqlclient
      sqlskymatch
      sqlupdate
      taplint
      tapquery
      tapresume
      tapskymatch
      tcat
      tcatn
      tcopy
      tcube
      tgridmap
      tjoin
      tloop
      tmatch1
      tmatch2
      tmatchn
      tmulti
      tmultin
      tpipe
      tskymap
      tskymatch2
      votcopy
      votlint
      xsdvalidate
      plot2plane
      plot2sky
      plot2cube
      plot2sphere
      plot2time
```

For help on the individual tasks, including their parameter lists, you can supply the word `help` after the task name, so for instance

```
stilts tcopy help
```

results in

```
Usage: tcopy ifmt=<in-format> ofmt=<out-format>
             [in=]<table> [out=]<out-table>
```

Finally, you can get help on any of the parameters of a task by writing `help=<param-name>`, like this:

```
stilts tcopy help=in
```

gives

```
Help for parameter IN in task TCOPY
-----------------------------------

   Name:
      in
```

```
      Usage:
         [in=]<table>

      Summary:
         Location of input table

      Description:
         The location of the input table. This may take one of the following
         forms:

          * A filename.
          * A URL.
          * The special value "-", meaning standard input. In this case the
               input format must be given explicitly using the ifmt parameter.
               Note that not all formats can be streamed in this way.
          * A scheme specification of the form :<scheme-name>:<scheme-args>.
          * A system command line with either a "<" character at the start, or
               a "|" character at the end ("<syscmd" or "syscmd|"). This
               executes the given pipeline and reads from its standard output.
               This will probably only work on unix-like systems.

         In any case, compressed data in one of the supported compression
         formats (gzip, Unix compress or bzip2) will be decompressed
         transparently.

      Type:
         uk.ac.starlink.table.StarTable
```

If you use "*" instead of a parameter name in this usage, help for all the parameters will be printed. Note that in most shells you will probably need to quote the asterisk, so you should write

```
stilts tcopy help='*'
```

In some cases, as described in Section 2.3, you will be prompted for the value of a parameter with a line something like this:

```
matcher - Name of matching algorithm [sky]:
```

In this case, if you enter "help" or a question mark, then the parameter help entry will be printed to the screen, and the prompt will be repeated.

For more detailed descriptions of the tasks, which includes explanatory comments and examples as well as the information above, see the full task descriptions in the Command Reference (Appendix B).

3 Invocation

There are a number of ways of invoking commands in the `stilts` application, depending on how you have installed the package. This section describes how to invoke it from the command line. Other options are using it from Jython (the Java implementation of the Python language) as described in Section 4, invoking it over HTTP as described in Section 11, and invoking it from within a Java application as described in Section 12.

If you're using a Unix-like operating system, the easiest way is to use the `stilts` script. If you have a full starjava installation it is in the `starjava/bin` directory. Otherwise you can download it separately from wherever you got your STILTS installation in the first place, or find it at the top of the `stilts.jar` or `topcat-*.jar` that contains your STILTS installation, so do something like

```
unzip stilts.jar stilts
chmod +x stilts
```

to extract it (if you don't have `unzip`, try `jar xvf stilts.jar stilts`). `stilts` is a simple shell script which just invokes java with the right classpath and the supplied arguments.

To run using the `stilts` script, first make sure that both the `java` executable and the `stilts` script itself are on your path, and that the `stilts.jar` or `topcat-*.jar` jar file is in the same directory as `stilts`. Then the form of invocation is:

```
stilts <java-flags> <stilts-flags> <task-name> <task-args>
```

A simple example would be:

```
stilts votcopy format=binary t1.xml t2.xml
```

in this case, as often, there are no `<java-flags>` or `<stilts-flags>`. If you use the `-classpath` argument or have a CLASSPATH environment variable set, then classpath elements thus specified will be added to the classpath required to run the command. The examples in the command descriptions below use this form for convenience.

If you don't have a Unix-like shell available however, you will need to invoke Java directly with the appropriate classes on your classpath. If you have the file `stilts.jar`, in most cases you can just write:

```
java <java-flags> -jar stilts.jar <stilts-flags> <task-name> <task-args>
```

which in practice would look something like

```
java -jar /some/where/stilts.jar votcopy format=binary t1.xml t2.xml
```

In the most general case, Java's `-jar` flag might be no good, for one of the following reasons:

1. You have the classes in some form other than the `stilts.jar` file (such as `topcat-full.jar`)
2. You need to specify some extra classes on the classpath, which is required e.g. for use with JDBC (Section 3.4) or if you are extending the commands (Section 10.8.3) using your own classes at runtime

In this case, you will need an invocation of this form:

```
java <java-flags> -classpath <class-path>
     uk.ac.starlink.ttools.Stilts <stilts-flags> <task-name> <task-args>
```

The example above in this case would look something like:

```
java -classpath /some/where/topcat-full.jar uk.ac.starlink.ttools.Stilts
     votcopy format=binary t1.xml t2.xml
```

Finally, as a convenience, it is possible to run STILTS from a TOPCAT installation by using its `-stilts` flag, like this:

```
topcat <java-flags> -stilts <stilts-flags> <task-name> <task-args>
```

This is possible because TOPCAT is built on top of STILTS, so contains a superset of its code.

The `<stilts-flags>`, `<task-name>` and `<task-args>` parts of these invocations are explained in Section 2, and the `<class-path>` and `<java-flags>` parts are explained in the following subsections.

3.1 Class Path

The classpath is the list of places that Java looks to find the bits of compiled code that it uses to run an application. Depending on how you have done your installation the core STILTS classes could be in various places, but they are probably in a file with one of the names `stilts.jar`, `topcat-lite.jar` or `topcat-full.jar`. The full pathname of one of these files can therefore be used as your classpath. In some cases these files are self-contained and in some cases they reference other jar files in the filesystem - this means that they may or may not continue to work if you move them from their original location.

Under certain circumstances the tools might need additional classes, for instance:

- JDBC drivers (see Section 3.4)
- Providing extended algebraic functions (see Section 10.8.3)
- Installing I/O handlers for new table formats (see SUN/252)

In this case the classpath must contain a list of all the jar files in which the required classes can be found, separated by colons (unix) or semicolons (MS Windows). Note that even if all your jar files are in a single directory you can't use the name of that directory as a class path - you must name each jar file, separated by colons/semicolons.

3.2 Java Flags

In most cases it is not necessary to specify any additional arguments to the Java runtime, but it can be useful in certain circumstances. The two main kinds of options you might want to specify directly to Java are these:

System properties

System properties are a way of getting information into the Java runtime from the outside, rather like environment variables. There is a list of the ones which have significance to STILTS in Section 3.3. You can set them from the command line using a flag of the form `-Dname=value`. So for instance to ensure that temporary files are written to the `/home/scratch` directory, you could use the flag

```
-Djava.io.tmpdir=/home/scratch
```

Memory size

Java runs with a fixed amount of 'heap' memory; this is typically 64Mb by default. If one of the tools fails with a message that says it's out of memory then this has proved too small for the job in hand. You can increase the heap memory with the `-Xmx` flag. To set the heap memory size to 256 megabytes, use the flag

```
-Xmx256M
```

(don't forget the 'M' for megabyte). You will probably find performance is dreadful if you specify a heap size larger than the physical memory of the machine you're running on.

You can specify other options to Java such as tuning and profiling flags etc, but if you want to do that sort of thing you probably don't need me to tell you about it.

If you are using the `stilts` command-line script, any flags to it starting `-D` or `-X` are passed directly to the `java` executable. You can pass other flags to Java with the `stilts` script's `-J` flag; for instance:

```
stilts -Xmx4M -J-verbose:gc calc 'mjdToIso(0)'
```

is equivalent to

```
java -Xmx4M -verbose:gc -jar stilts.jar calc 'mjdToIso(0)'
```

3.3 System Properties

System properties are a way of getting information into the Java runtime - they are a bit like environment variables. There are two ways to set them when using STILTS: either on the command line using arguments of the form `-Dname=value` (see Section 3.2) or in a file in your home directory named `.starjava.properties`, in the form of a `name=value` line. Thus submitting the flag

```
-Dvotable.strict=false
```

on the command line is equivalent to having the following in your `.starjava.properties` file:

```
#  Force strict interpretation of the VOTable standard.
votable.strict=false
```

The following system properties have special significance to STILTS:

`http.proxyHost`
Can be used to force HTTP access to go via a named proxy; may be required if you are attempting access to remote data or services from behind a firewall configured to block direct HTTP connections. See java documentation for this property for more details.

`java.awt.headless`
May need to be set to "`true`" if running the plotting tasks on a headless server. You only need to worry about this if you see error messages complaining about headlessness.

`java.io.tmpdir`
The directory in which STILTS will write any temporary files it needs. This defaults to the system temporary directory (e.g. `/tmp` on Unix), so if working with large unmapped (e.g. CSV) tables on a machine with limited space on the default disk, it may be necessary to change it.

`java.util.concurrent.ForkJoinPool.common.parallelism`
Controls the level of parallelisation done by certain processing, currently mainly visualisation. By default it is typically set to one less than the number of processing cores on the current machine. To inhibit parallelisation (e.g. if you suspect that the parallel output is giving different results to sequential processing) you can set this to 1.

`jdbc.drivers`
Can be set to a (colon-separated) list of JDBC driver classes using which SQL databases can be accessed (see Section 3.4).

`jel.classes`
Can be set to a (colon-separated) list of classes containing static methods which define user-provided functions for synthetic columns or subsets. (see Section 10.8.3).

`mark.workaround`
If set to "true", this will work around a bug in the `mark()`/`reset()` methods of some java `InputStream` classes. These are rather common, including in Sun's J2SE system libraries. Use this if you are seeing errors that say something like "`Resetting to invalid mark`". Currently defaults to "false".

`service.maxparallel`
Raises the maximum number of concurrent queries that may be made during a multi-cone operation. You should only increase this value **with great care** since you risk overloading servers and becoming unpopular with data centres. As a rule, you should only increase this value if you have obtained permission from the data centres whose services on which you will be using the increased parallelism.

`star.basicauth.user`
`star.basicauth.password`
If set, these will provide username and password for HTTP Basic Authentication. Any time the application attempts to access an HTTP URL and is met by a 401 Unauthorized response, it will try again supplying these user credentials. This is a rather blunt instrument, since the same identity is supplied regardless of which URL is being accessed, but it may be of some use in accessing basic-authentication protected services.

`startable.readers`
Can be set to a (colon-separated) list of custom table format input handler classes (see SUN/252). Each class must implement the `uk.ac.starlink.table.TableBuilder` interface, and must have a no-arg constructor. The readers thus named will be available alongside the standard ones listed in Section 5.1.1.

`startable.schemes`
Can be set to a (colon-separated) list of custom table scheme handler classes. Each class must implement the `uk.ac.starlink.table.TableScheme` interface, and must have a no-arg constructor. The schemes thus named will be available alongside the standard ones listed in Section 5.3.

`startable.storage`
Can be set to determine the default storage policy. Setting it to "`disk`" has basically the same effect as supplying the "`-disk`" argument on the command line (see Section 2.1). Other possible values are "`adaptive`", "`memory`", "`sideways`" and "`discard`"; see SUN/252. The default is "`adaptive`", which means storing smaller tables in memory, and larger ones on disk.

`startable.unmap`
Determines whether and how unmapping of memory mapped buffers is done. Possible values are "`sun`" (the default), "`cleaner`", "`unsafe`" or "`none`". In most cases you are advised to leave this alone, but in the event of unmapping-related JVM crashes (not expected!), setting it to `none` may help.

`startable.writers`
Can be set to a (colon-separated) list of custom table format output handler classes (see SUN/252). Each class must implement the `uk.ac.starlink.table.StarTableWriter` interface, and must have a no-arg constructor. The writers thus named will be available alongside the standard ones listed in Section 5.1.2.

`votable.namespacing`
Determines how namespacing is handled in input VOTable documents. Known values are "`none`" (no namespacing, xmlns declarations in VOTable document will probably confuse parser), "`lax`" (anything that looks like it is probably a VOTable element will be treated as a VOTable element) and "`strict`" (VOTable elements must be properly declared in one of the correct VOTable namespaces). May also be set to the classname of a `uk.ac.starlink.votable.Namespacing` implementation. The default is "`lax`".

`votable.strict`

Controls the behaviour when encountering a VOTable `FIELD` or `PARAM` element with a `datatype` attribute of `char`/`unicodeChar`, and no `arraysize` attribute. The VOTable standard says this indicates a single character, but some VOTables omit arraysize specification by accident when they intend `arraysize="*"`. If `votable.strict` is set `true`, a missing arraysize will be interpreted as meaning a single character, and if `false`, it will be interpreted as a variable-length array of characters (a string). The default is `true`.

`votable.version`
Selects the version of the VOTable standard which output VOTables will conform to by default. May take the values "`1.0`", "`1.1`", "`1.2`", "`1.3`" or "`1.4`". By default, version 1.4 VOTables are written.

3.4 JDBC Configuration

This section describes additional configuration which must be done to allow the commands to access SQL-compatible relational databases for reading or writing tables. If you don't need to talk to SQL-type databases, you can ignore the rest of this section. The steps described here are the standard ones for configuring JDBC (which sort-of stands for Java Database Connectivity); you can find more information on that on the web. The best place to look may be within the documentation of the RDBMS you are using.

To use STILTS with SQL-compatible databases you must:

- Have access to an SQL-compatible database locally or over the network
- Have a JDBC driver appropriate for that database
- Install that driver for use with STILTS
- Know the format the driver uses for URLs to access database tables
- Have appropriate privileges on the database to perform the desired operations

Installing the driver consists of two steps:

1. Ensure that the classpath you are using includes this driver class as described in Section 3.1
2. Set the `jdbc.drivers` system property to the name of the driver class as described in Section 3.3

Here is an example of using `tpipe` to write the results of an SQL query on a table in a MySQL database as a VOTable:

```
stilts -classpath /usr/local/jars/mysql-connector-java.jar \
       -Djdbc.drivers=com.mysql.jdbc.Driver \
       tpipe \
       in="jdbc:mysql://localhost/db1#SELECT id, ra, dec FROM gsc WHERE mag < 9" \
       ofmt=votable gsc.vot
```

or invoking Java directly:

```
java -classpath stilts.jar:/usr/local/jars/mysql-connect-java.jar \
     -Djdbc.drivers=com.mysql.jdbc.Driver \
     uk.ac.starlink.ttools.Stilts tpipe \
     in="jdbc:mysql://localhost/db1#SELECT id, ra, dec FROM gsc WHERE mag < 9" \
     ofmt=votable out=gsc.vot
```

You have to exercise some care to get the arguments in the right order here - see Section 3.

Alternatively, you can set some of this up beforehand to make the invocation easier. If you set your CLASSPATH environment variable to include the driver jar file (and the STILTS classes if you're invoking Java directly rather than using the scripts), and if you put the line

```
jdbc.drivers=com.mysql.jdbc.Driver
```

in the `.starjava.properties` file in your home directory, then you could avoid having to give the `-classpath` and `-Djdbc.drivers` flags respectively.

Below are presented the results of some experiments with JDBC drivers. Note however that **this information may be be incomplete and out of date**. If you have updates, feel free to pass them on and they may be incorporated here.

To the author's knowledge, STILTS has successfully been used with the following RDBMSs and corresponding JDBC drivers:

MySQL
MySQL has been tested on Linux with the Connector/J driver and seems to work; tested versions are server 3.23.55 with driver 3.0.8 and server 4.1.20 with driver 5.0.4. Sometimes tables with very many (hundreds of) columns cannot be written owing to SQL statement length restrictions. Note there is known to be a column metadata bug in version 3.0.6 of the driver which can cause a ClassCastException error when tables are written. Check the driver's documentation for additional parameters, for instance "`useUnicode=true&characterEncoding=UTF8`" may be required to handle some non-ASCII characters.

PostgreSQL
PostgreSQL 7.4.1 apparently works with its own driver. Note the performance of this driver appears to be rather poor, at least for writing tables.

Oracle
You can use Oracle with the JDBC driver that comes as part of its Basic Instant Client Package.

SQL Server
There is more than one JDBC driver known to work with SQL Server, including jTDS and its own JDBC driver. Some evidence suggests that jTDS may be the better choice, but your mileage may vary.

Sybase ASE
There has been a successful use of Sybase 12.5.2 and jConnect (jconn3.jar) using a JDBC URL like "`jdbc:sybase:Tds:hostname:port/dbname?user=XXX&password=XXX#SELECT...`". An earlier attempt using Sybase ASE 11.9.2 failed.

It is probably possible to use other RDBMSs and drivers, but you may have to do some homework.

Here are some example command lines that at least have at some point got STILTS running with databases:

PostgreSQL

```
stilts -classpath pg73jdbc3.jar \
       -Djdbc.drivers=org.postgresql.Driver ...
```

MySQL

```
stilts -classpath mysql-connector-java-3.0.8-bin.jar \
       -Djdbc.drivers=com.mysql.jdbc.Driver ...
```

Oracle

```
stilts -classpath ojdbc14.jar \
       -Djdbc.drivers=oracle.jdbc.driver.OracleDriver ...
```

SQL Server with jTDS

```
stilts -classpath jtds-1.1.jar \
       -Djdbc.drivers=net.sourceforge.jtds.jdbc.Driver ...
```

4 JyStilts - STILTS from Python

Most of the discussions and examples in this document describe using STILTS as a standalone java application from the command line; in this case, scripting can be achieved by executing one STILTS command, followed by another, followed by another, perhaps controlled from a shell script, with intermediate results stored in files.

However, it is also possible to invoke STILTS commands from within the Jython environment. Jython is a pure-java implementation of the widely-used Python scripting language. Using Jython is almost exactly the same as using the more usual C-based Python, except that it is not possible to use extensions which use C code. This means that if you are familiar with Python programming, it is very easy to string STILTS commands together in Jython.

This approach has several advantages over the conventional command-line usage:

- You can make use of python programming constructions like loops, functions and variables
- Python syntax can be used to put together parameter values (especially referencing quoted strings or values containing embedded spaces) in a way which is often less painful than doing it from the shell
- Intermediate processing stages can be kept in memory (in a python variable) rather than having to write them out to a file and read them in for the next command; this can be much more efficient
- Because of the previous point, there are separate read, filter, processing and write commands, which means command lines can be shorter and less confusing
- The java startup overhead (typically a couple of seconds) happens only once when entering jython, not once for every STILTS command

Note however that you will *not* be able to introduce JyStilts commands into your larger existing Python programs if those rely on C-based extensions, such as NumPy and SciPy, since JyStilts will only run in JPython, while C-based extensions will only run in CPython. (See however JNumeric for some of the Numpy functionality from Jython.)

Usage from jython has syntax which is similar to command-line STILTS, but with a few changes. The following functions are defined by JyStilts:

- A function `tread`, which reads a table from a file or URL and turns it into a table object in jython
- A table method `write` which takes a table object and writes it to file
- A table method for each STILTS filter (e.g. `cmd_head`, `cmd_select`, `cmd_addcol`)
- A table method for each STILTS output mode (e.g. `mode_out`, `mode_meta`, `mode_samp`),
- A function for each STILTS task (e.g. `tmatch2`, `tcat`, `plot2sky`)
- A number of table methods which make table objects integrate nicely into the python environment

Reasonably detailed documentation for these is provided in the usual Python way ("*doc strings*"), and can be accessed using the Python "`help`" command, however for full documentation and examples you should refer to this document.

In JyStilts the input, processing, filtering and output are done in separate steps, unlike in command-line STILTS where they all have to be combined into a single line. This can make the flow of execution easier to follow. A typical sequence will involve:

1. Reading one or more tables from file using the `tread` function
2. Perhaps filtering the input table(s) using one or more of the `cmd_*` filter methods
3. Performing core processing such as crossmatching
4. Perhaps filtering the result using one or more of the `cmd_*` filter methods
5. If running interactively, perhaps examining the intermediate results using one of the `mode_*`

output modes
6. Writing the final result to a file using the `write` method

Here is an example command line invocation for crossmatching two tables:

```
stilts tskymatch2 in1=survey.fits \
             icmd1='addskycoords fk4 fk5 RA1950 DEC1950 RA2000 DEC2000' \
             in2=mycat.csv ifmt2=csv \
             icmd2='select VMAG>18' \
             ra1=ALPHA dec1=DELTA ra2=RA2000 dec2=DEC2000 \
             error=10 join=2not1 \
             out=matched.fits
```

and here is what it might look like in JyStilts:

```
>>> import stilts
>>> t1 = stilts.tread('survey.fits')
>>> t1 = t1.cmd_addskycoords(t1, 'fk4', 'fk5', 'RA1950', 'DEC1950', 'RA2000', 'DEC2000')
>>> t2 = stilts.tread('mycat.csv', 'csv')
>>> t2 = t2.cmd_select('VMAG>18')
>>> tm = stilts.tskymatch2(in1=t1, in2=t2, ra1='ALPHA', dec1='DELTA',
...                        error=10, join='2not1')
>>> tm.write('matched.fits')
```

When running interactively, it can be convenient to examine the intermediate results before processing or writing as well, for instance:

```
>>> tm.mode_count()
columns: 19   rows: 2102
>>> tm.cmd_keepcols('ID ALPHA DELTA').cmd_head(4).write()
+--------+---------------+-----------+
| ID     | ALPHA         | DELTA     |
+--------+---------------+-----------+
| 262    | 149.82439     | -0.11249  |
| 263    | 150.14438     | -0.11785  |
| 265    | 149.92944     | -0.11667  |
| 273    | 149.93185     | -0.12566  |
+--------+---------------+-----------+
```

More detail about how to run JyStilts and its usage is given in the following subsections.

4.1 Running JyStilts

The easiest way to run JyStilts is to download the standalone `jystilts.jar` file from the STILTS web page, and simply run

```
java -jar jystilts.jar
```

This file includes jython itself and all the STILTS and JyStilts classes. To use the JyStilts commands, you will need to import the stilts module using a line like "`import stilts`" from Jython in the usual Python way.

Alternatively, you can run JyStilts from an existing Jython installation using just the `stilts.jar` file. First, make sure that Jython is installed; it is available from http://www.jython.org/, and comes as a self-installing jar file. JyStilts has been tested, and appears to work, with jython version 2.7.2. It also works with jython 2.5.* under Java 8 and Java 11, but jystilts with jython 2.5.* and Java 17 can fail with security problems. Some earlier versions of JyStilts worked with jython 2.2.1, but that no longer seems to be the case; it might be possible to reinstate this if there is some pressing need.

To use JyStilts, you then just need to start jython with the `stilts.jar` file on your classpath, for instance like this:

```
jython -J-classpath /some/where/stilts.jar
```

or (C-shell):

```
setenv CLASSPATH /some/where/stilts.jar
jython
```

Optionally, you can extract the `stilts.py` module from the stilts.jar file (using a command like "`unzip stilts.jar stilts.py`") and put it in a directory on your jython `sys.path` (e.g. `jythondir/Lib`); this may cause jython to compile it to bytecode (`stilts$py.class`) and thus improve startup time. Note that in this case you will still need the `stilts.jar` file on your classpath as above.

4.2 Table I/O

The `tread` function reads tables from an external location into JyStilts. Its arguments are as follows:

```
tread(location, fmt='(auto)', random=False)
```

and its return value is a table object, which can be interrogated directly, or used in other JyStilts commands. Usually, the location argument should be a string which gives the filename or URL at which a table can be found. You can alternatively use a readable python file (or file-like) object for the location, but be aware that this may be less efficient on memory. As with command-line STILTS, the `fmt` argument is one of the options in Section 5.1.1, but may be left as the default if the format auto-detectable, which currently means if the file is in VOTable, FITS, CDF, ECSV, PDS4, Parquet, Feather or GBIN format. The `random` argument can be used to ensure that the returned file has random (i.e. not sequential-only) access; for some table formats the default way of reading them in means that their rows can only be accessed in sequence. Depending on what processing you are doing, that may or may not be satisfactory.

Examples of reading a table are:

```
>>> import stilts
>>> t1 = stilts.tread('cat.fits')
>>> t2 = stilts.tread(open('cat.fits', 'rb'))           # less efficient
>>> t3 = stilts.tread('data.csv', fmt='csv', random=True)
```

The most straightforward way to write a table (presumably the result of one or a sequence of JyStilts commands) is using the `write` table method:

```
write(self, location=None, fmt='(auto)')
```

The `location` gives either a string which is a filename, or a writable python file (or file-like) object. Again, use of a filename is preferred as it may(?) be more efficient. If no location is supplied, the table will be written to standard output (useful for inspection, but a bad idea for binary formats or very large tables). The `fmt` argument is one of the output formats in Section 5.1.2, but may be left as the default if the format can be guessed from the filename.

Examples of writing a table are:

```
>>> table.write('out.fits')
>>> table.write(open('out.fits', 'wb'))       #  less efficient?
>>> table.write('catalogue.dat', fmt='csv')
>>> table.write()                             #  display to stdout
```

Often it's convenient to combine examining the table with filtering steps, for instance:

```
>>> table.every(100).write()
```

would write only every hundredth row, and

```
>>> (table.cmd_sorthead(10, 'BMAG')
...       .cmd_select('!NULL_VMAG')
...       .cmd_keepcols('BMAG VMAG')
...       .write())
```

would write only the BMAG and VMAG columns for the ten rows in which VMAG is non-null with the lowest BMAG values.

You can also read and write multiple tables, if you use a table format for which that is appropriate. This generally means FITS (which can store tables in multiple extensions) or VOTable (which can store multiple TABLE elements in one document). This is done using the `treads` and `twrites` functions. The functions look like this:

```
treads(location, fmt='(auto)', random=False)
twrites(tables, location=None, fmt='(auto)')
```

These are similar to the `tread` and `twrite` functions, except that `treads` returns a list of tables rather than a single table, and `twrites`'s `tables` argument is an iterable over tables rather than a single table. Here is an example of reading multiple tables from a multi-extension FITS file, counting the rows in each, and then writing them out to a multi-TABLE VOTable file:

```
import stilts
tables = stilts.treads('multi.fits')
print([t.getRowCount() for t in tables])
stilts.twrites(tables, 'multi.vot', fmt='votable')
```

4.3 Table objects

The tables read by the `tread` function and produced by operating on them within JyStilts have a number of methods defined on them. These are explained below.

First, a number of special methods are defined which allow a table to behave in python like a sequence of rows:

`__iter__`
: This special method means that the table can be treated as an *iterable*, so that for instance "`for row in table:`" will iterate over all rows.

`__len__` ***(random-access tables only)***
: This special method means that you can use the expression "`len(table)`" to count the number of rows. This method is not available for tables with sequential access only.

`__getitem__` ***(random-access tables only)***
: Returns a row at a given index in the table. This special method means that you can use indexing expressions like "`table[3]`" or `table[0:10]` to obtain the row or rows corresponding to a given row index or slice. This method is not available for tables with sequential access only.

`__add__`, `__mul__`, `__rmul__`
: These special methods allow the addition and multiplication operators "`+`" and and "`*`" to be used with the sense of concatenation. Thus "`table1+table2`" will produce a new table with the rows of `table1` followed by the rows of `table2`. Note this will only work if both tables have compatible columns. Similarly "`table*3`" would produce a table like `table` but with all its rows repeated three times.

In all of these cases, each row object that is accessed is a tuple of the column values for that row of the table. The tuple items (table cells) may be accessed using a key which is a numeric index or

slice in the usual way, or with a key which is a column name, or one of the ColumnInfo objects returned by `columns()`.

Sometimes, the result of a table operation will be a table which does not have random access. For such tables you can iterate over the rows, but not get their row values by indexing. Non-random-access tables are also peculiar in that `getRowCount` returns a negative value. To take a table which may not have random access and make it capable of random access, use the `random` filter: "`table=table.cmd_random()`".

To a large extent it is possible to duplicate the functions of the various STILTS commands by writing your own python code based on these python-friendly table access methods. Note however that such python-based processing is likely to be *much* slower than the STILTS equivalents. If performance is important to you, you should try in most cases to use the various `cmd_*` commands etc for table processing.

Second, some additional utility methods are defined:

`count_rows()`
: Returns the number of rows in the table in the most efficient way possible. If the table is random-access or otherwise knows its row count without further calculation, that value is returned. Otherwise, the rows are iterated over without reading, which may take some time but should be much more efficient than iterating over the table as an iterable, since the row cell data itself is not retrieved.

`columns()`
: Returns a tuple of the column descriptors for the table. Each item in the tuple is an instance of the ColumnInfo class; useful methods include `getName()`, `getUnitString()`, `getUCD()`. `str(column)` will return its name.

`coldata(key)`
: Returns a sequence of the values for the given column. The sequence will have the same number of elements as the number of rows in the table. The `key` argument may be either an integer column index (if negative, counts backwards from the end), or the column name or info object. The returned value will always be iterable (has `__iter__`), but will only be indexable (has `__len__` and `__getitem__`) if the table is random access.

`parameters()`
: Returns a name to value mapping of the table parameters (per-table metadata). This does not include all the available information about those parameters, for instance unit and UCD information is not included. For more detailed information, use the `StarTable` methods. Note that as currently implemented, changing the values in the returned mapping will not change the actual table parameter values.

`write(location=None, fmt=None)`
: Outputs the table. The optional `location` argument gives a filename or writable file object, and the optional `fmt` argument gives a format, one of the options listed in Section 5.1.1. If `location` is not supplied, output is to standard output, so in an interactive session it will be printed to the terminal. If `fmt` is not supplied, an attempt will be made to guess a suitable format based on the location.

Third, a set of `cmd_*` methods corresponding to the STILTS filters are available; these are described in Section 4.4.

Fourth, a set of `mode_*` methods corresponding to the STILTS output modes are available; these are described in Section 4.5.

Finally, tables are also instances of the StarTable interface defined by STIL, which is the table I/O

layer underlying STILTS. The full documentation can be found in the user manual and javadocs on the STIL page, and all the java methods can be used from JyStilts, but in most cases there are more pythonic equivalents provided, as described above.

Here are some examples of these methods in use:

```
>>> import stilts
>>> xsc = stilts.tread('/data/table/2mass_xsc.xml')  # read table
>>> xsc.mode_count()                                 # show rows/column count
columns: 6   rows: 1646844
>>> print xsc.columns()                              # full info on columns
(id(String), ra(Double)/degrees, dec(Double)/degrees, jmag(Double)/mag, hmag(Double)/mag, k
>>> print [str(col) for col in xsc.columns()]        # column names only
['id', 'ra', 'dec', 'jmag', 'hmag', 'kmag']
>>> row = xsc[1000000]                               # examine millionth row
>>> print row
(u'19433000+4003190', 295.875, 40.055286, 14.449, 13.906, 13.374)
>>> print row[0]                                     # cell by index
19433000+4003190
>>> print row['ra'], row['dec']                      # cells by col name
295.875 40.055286
>>> print len(xsc)                                   # count rows, maybe slow
1646844
>>> print xsc.count_rows()                           # count rows efficiently
1646844L
>>> print (xsc+xsc).count_rows()                     # concatenate
3293688L
>>> print (xsc*10000).count_rows()
16468440000L
>>> for row in xsc:                    # select rows using python commands
...     if row[4] - row[3] > 3.0:
...         print row[0]
...
11165243+2925509
20491597+5119089
04330238+0858101
01182715-1013248
11244075+5218078
>>>                                    # same thing using stilts (50x faster)
>>> (xsc.cmd_select('hmag - jmag > 3.0')
...     .cmd_keepcols('id')
...     .write())
+------------------+
| id               |
+------------------+
| 11165243+2925509 |
| 20491597+5119089 |
| 04330238+0858101 |
| 01182715-1013248 |
| 11244075+5218078 |
+------------------+
```

The following are all ways to obtain the value of a given cell in the table from the previous example.

```
xsc.getCell(99, 0)
xsc[99][0]
xsc[99]['id']
xsc.coldata(0)[99]
xsc.coldata('id')[99]
```

Some of these methods may be more efficient than others. Note that none of these methods will work if the table has sequential-only access.

4.4 Table filter commands (`cmd_*`)

The STILTS table filters documented in Section 6.1 are available in JyStilts as table methods which start with the "`cmd_`" prefix. The return value when calling the method on a table object is another

table object. The arguments, which are the same as those required for the command-line version, are supplied as a list of unnamed arguments of the `cmd_*` function. In general the arguments are strings, but numbers are accepted where appropriate. Use the python `help` command to see the usage of each method.

So, to use the `tail` filter to select only the last ten lines of a table, you can write:

```
table.cmd_tail(10)
```

To set units of "Hz" for some columns using the `colmeta` filter write:

```
table.cmd_colmeta('-units', 'Hz', 'AFREQ BFREQ CFREQ')
```

Note that where a filter argument is a space-separated list it must appear as a single argument in the filter invocation, just as in command-line STILTS.

The filter commands are also available as module functions. This means that

```
stilts.cmd_head(table, 10)
```

and

```
table.cmd_head(10)
```

have exactly the same meaning. It's a matter of taste which you prefer.

4.5 Table output modes (`mode_*`)

The STILTS table output modes documented in Section 6.4 are available in JyStilts as table methods which start with the "`mode_`" prefix. These methods have no return value, but cause something to happen, in some cases output to be written to standard output. Some of these methods have named arguments, others have no arguments. Use the python `help` command to see the usage of each method.

These methods are straightforward to use. The following example calculates statistics for a table and writes the results to standard output:

```
>>> table.mode_stats()
```

and this one attempts to send the table via the SAMP communications protocol to a running instance of TOPCAT:

```
>>> table.mode_samp(client='topcat')
```

The output modes are also available as module functions. This means that

```
stilts.mode_samp(table, client='topcat')
```

and

```
table.mode_samp(client='topcat)
```

have exactly the same meaning. It's a matter of taste which you prefer.

4.6 Tasks

The STILTS tasks documented in Appendix B can be used under their usual names if they are imported from the `stilts` module. STILTS parameters as are supplied as named arguments of the python functions. In general they are either table objects for table input parameters or strings, but in

some cases python arrays are accepted, and numbers may be used where appropriate. The STILTS input format (`ifmt`, `istream`), filter (`cmd`/`icmd`/`ocmd`) and output mode (`omode`) parameters are not used however; instead perform filtering directly on the table inputs and outputs using the python `cmd_*` and `mode_*` table methods or functions.

Here is an example of concatenating two similar tables together and writing the result:

```
>>> from stilts import tread, tcat
>>> t1 = tread('data1.csv', fmt='csv')
>>> t2 = tread('data2.csv', fmt='csv')
>>> t12 = tcat([t1,t2], seqcol='seq')
>>> t12.write('t12.csv', fmt='csv')
```

Note that for those tasks which have a parameter named "`in`" in command-line STILTS, it has been renamed as "`in_`" for the python version, to avoid a name clash with the python reserved word. In most cases, the `in` parameter is the first, mandatory parameter in any case, and so can be referenced by position as in the previous example (we could have written "`tcat(in_=[t1,t2])`" instead).

4.7 Calculation Functions

The various functions from the expression language listed in Section 10.6 are available directly from JyStilts. Each of the subsections in that section is a class in the `stilts` module namespace, with unbound functions representing the functions.

This means you can use them like this:

```
>>> import stilts
>>> print stilts.Times.mjdToIso(54292)
2007-07-11T00:00:00
```

or like this:

```
>>> from stilts import CoordsDegrees
>>> dist = CoordsDegrees.skyDistanceDegrees(ra1, dec1, ra2, dec2)
```

5 Table I/O

Most of the tools in this package either read one or more tables as input, or write one or more tables as output, or both. This section explains what kind of tables the tools can read and write, and how you tell them where to find the tables to operate on.

In most cases input and output table specifications are given by parameters with the following names (or similar ones):

`in`
: Location of the input table; usually a filename or URL, but maybe some other source of data (see Section 5.2) or a *scheme specification* (see Section 5.3)

`ifmt`
: Format of the input table, as listed in Section 5.1.1, or blank for automatic format detection based on input content or name (but ignored for *scheme specifications*)

`out`
: Destination of the output table; usually a filename or "`-`"/blank for standard output

`ofmt`
: Format of the output table as listed in Section 5.1.2, or blank for automatic format detection based on output filename

5.1 Table Formats

The generic table commands in STILTS (currently `tpipe`, `tcopy`, `tmulti`, `tmultin`, `tcat`, `tcatn`, `tloop`, `tjoin`, `tgridmap`, `tcube`, `tmatch1`, `tmatch2`, `tmatchn`, `tskymap`, `tskymatch2`, `pixfoot`, `pixsample`, `plot2cube`, `plot2plane`, `plot2sky`, `plot2sphere`, `plot2time`, `plot2d`, `plot3d`, `plothist`, `cdsskymatch`, `cone`, `coneskymatch`, `sqlskymatch`, `tapquery`, `tapresume`, `tapskymatch` and `regquery`) have no native format for table storage, they can process data in a number of formats equally well. STIL has its own model of what a table consists of, which is basically:

- Some per-table metadata (parameters)
- A number of columns
- Some per-column metadata
- A number of rows, each containing one entry per column

Some table formats have better facilities for storing this sort of thing than others, and when performing conversions STILTS does its best to translate between them, but it can't perform the impossible: for instance there is nowhere in a Comma-Separated Values file to store descriptions of column units, so these will be lost when converting from VOTable to CSV formats.

The formats the package knows about are dependent on the input and output handlers currently installed. The ones installed by default are listed in the following subsections. More may be added in the future, and it is possible to install new ones at runtime - see the STIL documentation for details.

Some formats can be used to hold multiple tables in a single file, and others can only hold a single table per file.

5.1.1 Input Formats

Some of the tools in this package ask you to specify the format of input tables using the `ifmt` (or a similarly named) parameter. For some file formats (e.g. FITS, VOTable, CDF), the format can be automatically determined by looking at the file content, regardless of filename; for others (e.g. CSV

files with a ".csv" extension), STILTS may be able to use the filename as a hint to guess the format (the details of these rules are given in the format-specific subsections below). Otherwise, you have to supply the format using the `ifmt` parameter. It is always safe to specify the format explicitly; this will be slightly more efficient than auto-determination, and may lead to more helpful error messages in the case that the table can't be read correctly.

The available input formats are described in the following subsections. It is also possible to add new formats at runtime using the `startable.readers` system property, or by setting the format to the classname of a `uk.ac.starlink.table.TableBuilder` class.

5.1.1.1 `fits`

FITS is a very well-established format for storage of astronomical table or image data (see https://fits.gsfc.nasa.gov/). This reader can read tables stored in binary (`XTENSION='BINTABLE'`) and ASCII (`XTENSION='TABLE'`) table extensions; any image data is ignored. Currently, binary table extensions are read much more efficiently than ASCII ones.

When a table is stored in a BINTABLE extension in an uncompressed FITS file on disk, the table is 'mapped' into memory; this generally means very fast loading and low memory usage. FITS tables are thus usually efficient to use.

Limited support is provided for the semi-standard HEALPix-FITS convention; such information about HEALPix level and coordinate system is read and made available for application usage and user examination.

A private convention is used to support encoding of tables with more than 999 columns (not possible in standard FITS); this was discussed on the FITSBITS mailing list in July 2017 in the thread BINTABLE convention for >999 columns.

Header cards in the table's HDU header will be made available as table parameters. Only header cards which are not used to specify the table format itself are visible as parameters (e.g. NAXIS, TTYPE* etc cards are not). HISTORY and COMMENT cards are run together as one multi-line value.

Any 64-bit integer column with a non-zero integer offset (`TFORMn='K'`, `TSCALn=1`, `TZEROn<>0`) is represented in the read table as Strings giving the decimal integer value, since no numeric type in Java is capable of representing the whole range of possible inputs. Such columns are most commonly seen representing unsigned long values.

Where a multi-extension FITS file contains more than one table, a single table may be specified using the position indicator, which may take one of the following forms:

- The numeric index of the HDU. The first extension (first HDU after the primary HDU) is numbered 1. Thus in a compressed FITS table named "`spec23.fits.gz`" with one primary HDU and two BINTABLE extensions, you would view the first one using the name "`spec23.fits.gz`" or "`spec23.fits.gz#1`" and the second one using the name "`spec23.fits.gz#2`". The suffix "`#0`" is never used for a legal FITS file, since the primary HDU cannot contain a table.
- The name of the extension. This is the value of the `EXTNAME` header in the HDU, or alternatively the value of `EXTNAME` followed by "`-`" followed by the value of `EXTVER`. This follows the recommendation in the FITS standard that `EXTNAME` and `EXTVER` headers can be used to identify an HDU. So in a multi-extension FITS file "`cat.fits`" where a table extension has `EXTNAME='UV_DATA'` and `EXTVER=3`, it could be referenced as "`cat.fits#UV_DATA`" or "`cat.fits#UV_DATA-3`". Matching of these names is case-insensitive.

Files in this format may contain multiple tables; depending on the context, either one or all tables

will be read. Where only one table is required, either the first one in the file is used, or the required one can be specified after the "#" character at the end of the filename.

This format can be automatically identified by its content so you do not need to specify the format explicitly when reading FITS tables, regardless of the filename.

There are actually two FITS input handlers, `fits-basic` and `fits-plus`. The `fits-basic` handler extracts standard column metadata from FITS headers of the HDU in which the table is found, while the `fits-plus` handler reads column and table metadata from VOTable content stored in the primary HDU of the multi-extension FITS file. FITS-plus is a private convention effectively defined by the corresponding output handler; it allows de/serialization of much richer metadata than can be stored in standard FITS headers when the FITS file is read by fits-plus-aware readers, though other readers can understand the unenhanced FITS file perfectly well. It is normally not necessary to worry about this distinction; STILTS will determine whether a FITS file is FITS-plus or not based on its content and use the appropriate handler, but if you want to force the reader to use or ignore the enriched header, you can explicitly specify an input format of "`fits-plus`" or "`fits-basic`".

5.1.1.2 `colfits`

As well as normal binary and ASCII FITS tables, STIL supports FITS files which contain tabular data stored in column-oriented format. This means that the table is stored in a BINTABLE extension HDU, but that BINTABLE has a single row, with each cell of that row holding a whole column's worth of data. The final (slowest-varying) dimension of each of these cells (declared via the `TDIMn` headers) is the same for every column, namely, the number of rows in the table that is represented. The point of this is that all the cells for each column are stored contiguously, which for very large, and especially very wide tables means that certain access patterns (basically, ones which access only a small proportion of the columns in a table) can be much more efficient since they require less I/O overhead in reading data blocks.

Such tables are perfectly legal FITS files, but general-purpose FITS software may not recognise them as multi-row tables in the usual way. This format is mostly intended for the case where you have a large table in some other format (possibly the result of an SQL query) and you wish to cache it in a way which can be read efficiently by a STIL-based application.

For performance reasons, it is advisable to access colfits files uncompressed on disk. Reading them from a remote URL, or in gzipped form, may be rather slow (in earlier versions it was not supported at all).

This format can be automatically identified by its content so you do not need to specify the format explicitly when reading colfits-basic tables, regardless of the filename.

Like the normal (row-oriented) FITS handler, two variants are supported: with (`colfits-plus`) or without (`colfits-basic`) metadata stored as a VOTable byte array in the primary HDU.

5.1.1.3 `votable`

VOTable is an XML-based format for tabular data endorsed by the International Virtual Observatory Alliance; while the tabular data which can be encoded is by design close to what FITS allows, it provides for much richer encoding of structure and metadata. Most of the table data exchanged by VO services is in VOTable format, and it can be used for local table storage as well.

Any table which conforms to the VOTable 1.0, 1.1, 1.2, 1.3 or 1.4 specifications can be read. This includes all the defined cell data serializations; cell data may be included in-line as XML elements (TABLEDATA serialization), included/referenced as a FITS table (FITS serialization), or

included/referenced as a raw binary stream (BINARY or BINARY2 serialization). The handler does not attempt to be fussy about input VOTable documents, and it will have a good go at reading VOTables which violate the standards in various ways.

Much, but not all, of the metadata contained in a VOTable document is retained when the table is read in. The attributes `unit`, `ucd`, `xtype` and `utype`, and the elements `COOSYS`, `TIMESYS` and `DESCRIPTION` attached to table columns or parameters, are read and may be used by the application as appropriate or examined by the user. However, information encoded in the hierarchical structure of the VOTable document, including `GROUP` structure, is not currently retained when a VOTable is read.

VOTable documents may contain more than one actual table (`TABLE` element). To specify a specific single table, the table position indicator is given by the zero-based index of the `TABLE` element in a breadth-first search. Here is an example VOTable document:

```
<VOTABLE>
  <RESOURCE>
    <TABLE name="Star Catalogue"> ... </TABLE>
    <TABLE name="Galaxy Catalogue"> ... </TABLE>
  </RESOURCE>
</VOTABLE>
```

If this is available in a file named "cats.xml" then the two tables could be named as "cats.xml#0" and "cats.xml#1" respectively.

Files in this format may contain multiple tables; depending on the context, either one or all tables will be read. Where only one table is required, either the first one in the file is used, or the required one can be specified after the "`#`" character at the end of the filename.

This format can be automatically identified by its content so you do not need to specify the format explicitly when reading VOTable tables, regardless of the filename.

5.1.1.4 `cdf`

NASA's Common Data Format, described at http://cdf.gsfc.nasa.gov/, is a binary format for storing self-described data. It is typically used to store tabular data for subject areas like space and solar physics.

This format can be automatically identified by its content so you do not need to specify the format explicitly when reading CDF tables, regardless of the filename.

5.1.1.5 `csv`

Comma-separated value ("CSV") format is a common semi-standard text-based format in which fields are delimited by commas. Spreadsheets and databases are often able to export data in some variant of it. The intention is to read tables in the version of the format spoken by MS Excel amongst other applications, though the documentation on which it was based was not obtained directly from Microsoft.

The rules for data which it understands are as follows:

- Each row must have the same number of comma-separated fields.
- Whitespace (space or tab) adjacent to a comma is ignored.
- Adjacent commas, or a comma at the start or end of a line (whitespace apart) indicates a null field.
- Lines are terminated by any sequence of carriage-return or newline characters ('\r' or '\n') (a corollary of this is that blank lines are ignored).

- Cells may be enclosed in double quotes; quoted values may contain linebreaks (or any other character); a double quote character within a quoted value is represented by two adjacent double quotes.
- The first line *may* be a header line containing column names rather than a row of data. Exactly the same syntactic rules are followed for such a row as for data rows.

Note that you can *not* use a "`#`" character (or anything else) to introduce "comment" lines.

Because the CSV format contains no metadata beyond column names, the handler is forced to guess the datatype of the values in each column. It does this by reading the whole file through once and guessing on the basis of what it has seen (though see the `maxSample` configuration option). This has the disadvantages:

- Sometimes it guesses a different type than what you want (e.g. 32-bit integer rather than 64-bit integer)
- It's slow to read.

This means that CSV is not generally recommended if you can use another format instead. If you're stuck with a large CSV file that's misbehaving or slow to use, one possibility is to turn it into an ECSV file file by adding some header lines by hand.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`csv(header=true,maxSample=100000)`". The following options are available:

`header = true|false|null`
: Indicates whether the input CSV file contains the optional one-line header giving column names. Options are:

  - `true`: the first line is a header line containing column names
  - `false`: all lines are data lines, and column names will be assigned automatically
  - `null`: a guess will be made about whether the first line is a header or not depending on what it looks like

  The default value is `null` (auto-determination). This usually works OK, but can get into trouble if all the columns look like string values.

`maxSample = <int>`
: Controls how many rows of the input file are sampled to determine column datatypes. When reading CSV files, since no type information is present in the input file, the handler has to look at the column data to see what type of value appears to be present in each column, before even starting to read the data in. By default it goes through the whole table when doing this, which can be time-consuming for large tables. If this value is set, it limits the number of rows that are sampled in this data characterisation pass, which can reduce read time substantially. However, if values near the end of the table differ in apparent type from those near the start, it can also result in getting the datatypes wrong.

This format cannot be automatically identified by its content, so in general it is necessary to specify that a table is in CSV format when reading it. However, if the input file has the extension "`.csv`" (case insensitive) an attempt will be made to read it using this format.

An example looks like this:

```
RECNO,SPECIES,NAME,LEGS,HEIGHT,MAMMAL
1,pig,Pigling Bland,4,0.8,true
2,cow,Daisy,4,2.0,true
3,goldfish,Dobbin,,0.05,false
4,ant,,6,0.001,false
5,ant,,6,0.001,false
6,queen ant,Ma'am,6,0.002,false
```

```
7,human,Mark,2,1.8,true
```

See also ECSV as a format which is similar and capable of storing more metadata.

5.1.1.6 ecsv

The Enhanced Character Separated Values format was developed within the Astropy project and is described in Astropy APE6 (DOI). It is composed of a YAML header followed by a CSV-like body, and is intended to be a human-readable and maybe even human-writable format with rich metadata. Most of the useful per-column and per-table metadata is preserved when de/serializing to this format. The version supported by this reader is currently ECSV 1.0.

There are various ways to format the YAML header, but a simple example of an ECSV file looks like this:

```
# %ECSV 1.0
# ---
# delimiter: ','
# datatype: [
#   { name: index,   datatype: int32   },
#   { name: Species, datatype: string  },
#   { name: Name,    datatype: string  },
#   { name: Legs,    datatype: int32   },
#   { name: Height,  datatype: float64, unit: m },
#   { name: Mammal,  datatype: bool    },
# ]
index,Species,Name,Legs,Height,Mammal
1,pig,Bland,4,,True
2,cow,Daisy,4,2,True
3,goldfish,Dobbin,,0.05,False
4,ant,,6,0.001,False
5,ant,,6,0.001,False
6,human,Mark,2,1.9,True
```

If you follow this pattern, it's possible to write your own ECSV files by taking an existing CSV file and decorating it with a header that gives column datatypes, and possibly other metadata such as units. This allows you to force the datatype of given columns (the CSV reader guesses datatype based on content, but can get it wrong) and it can also be read much more efficiently than a CSV file and its format can be detected automatically.

The header information can be provided either in the ECSV file itself, or alongside a plain CSV file from a separate source referenced using the `header` configuration option. In Gaia EDR3 for instance, the ECSV headers are supplied alongside the CSV files available for raw download of all tables in the Gaia source catalogue, so e.g. STILTS can read one of the gaia_source CSV files with full metadata as follows:

```
   stilts tpipe
      ifmt='ecsv(header=http://cdn.gea.esac.esa.int/Gaia/gedr3/ECSV_headers/gaia_source.header
      in=http://cdn.gea.esac.esa.int/Gaia/gedr3/gaia_source/GaiaSource_000000-003111.csv.gz
```

The ECSV datatypes that work well with this reader are `bool`, `int8`, `int16`, `int32`, `int64`, `float32`, `float64` and `string`. Array-valued columns are also supported with some restrictions. Following the ECSV 1.0 specification, columns representing arrays of the supported datatypes can be read, as columns with `datatype: string` and a suitable `subtype`, e.g. "`int32[<dims>]`" or "`float64[<dims>]`". Fixed-length arrays (e.g. `subtype: int32[3,10]`) and 1-dimensional variable-length arrays (e.g. `subtype: float64[null]`) are supported; however variable-length arrays with more than one dimension (e.g. `subtype: int32[4,null]`) cannot be represented, and are read in as string values. Null elements of array-valued cells are not supported; they are read as NaNs for floating point data, and as zero/false for integer/boolean data. ECSV 1.0, required to work with array-valued columns, is supported by Astropy v4.3 and later.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`ecsv(header=http://cdn.gea.esac.esa.int/Gaia/gedr3/ECSV_headers/gaia_source.header,colcheck=F`
The following options are available:

`header = <filename-or-url>`
Location of a file containing a header to be applied to the start of the input file. By using this you can apply your own ECSV-format metadata to plain CSV files.

`colcheck = IGNORE|WARN|FAIL`
Determines the action taken if the columns named in the YAML header differ from the columns named in the first line of the CSV part of the file.

This format can be automatically identified by its content so you do not need to specify the format explicitly when reading ECSV tables, regardless of the filename.

5.1.1.7 `ascii`

In many cases tables are stored in some sort of unstructured plain text format, with cells separated by spaces or some other delimiters. There is a wide variety of such formats depending on what delimiters are used, how columns are identified, whether blank values are permitted and so on. It is impossible to cope with them all, but the ASCII handler attempts to make a good guess about how to interpret a given ASCII file as a table, which in many cases is successful. In particular, if you just have columns of numbers separated by something that looks like spaces, you should be just fine.

Here are the detailed rules for how the ASCII-format tables are interpreted:

- Bytes in the file are interpreted as ASCII characters
- Each table row is represented by a single line of text
- Lines are terminated by one or more contiguous line termination characters: line feed (0x0A) or carriage return (0x0D)
- Within a line, fields are separated by one or more whitespace characters: space (" ") or tab (0x09)
- A field is either an unquoted sequence of non-whitespace characters, or a sequence of non-newline characters between matching single (') or double (") quote characters - spaces are therefore allowed in quoted fields
- Within a quoted field, whitespace characters are permitted and are treated literally
- Within a quoted field, any character preceded by a backslash character ("\") is treated literally. This allows quote characters to appear within a quoted string.
- An empty quoted string (two adjacent quotes) or the string "`null`" (unquoted) represents the null value
- All data lines must contain the same number of fields (this is the number of columns in the table)
- The data type of a column is guessed according to the fields that appear in the table. If all the fields in one column can be parsed as integers (or null values), then that column will turn into an integer-type column. The types that are tried, in order of preference, are: `Boolean`, `Short Integer`, `Long`, `Float`, `Double`, `String`
- Some special values are permitted for floating point columns: `NaN` for not-a-number, which is treated the same as a null value for most purposes, and `Infinity` or `inf` for infinity (with or without a preceding +/- sign). These values are matched case-insensitively.
- Empty lines are ignored
- Anything after a hash character "#" (except one in a quoted string) on a line is ignored as far as table data goes; any line which starts with a "!" is also ignored. However, lines which start with a "#" or "!" at the start of the table (before any data lines) will be interpreted as metadata as follows:

- The last "#"/"!"-starting line before the first data line may contain the column names. If it has the same number of fields as there are columns in the table, each field will be taken to be the title of the corresponding column. Otherwise, it will be taken as a normal comment line.
- Any comment lines before the first data line not covered by the above will be concatenated to form the "description" parameter of the table.

If the list of rules above looks frightening, don't worry, in many cases it ought to make sense of a table without you having to read the small print. Here is an example of a suitable ASCII-format table:

```
#
# Here is a list of some animals.
#
# RECNO  SPECIES         NAME          LEGS   HEIGHT/m
  1      pig             "Pigling Bland"  4  0.8
  2      cow             Daisy         4      2
  3      goldfish        Dobbin        ""     0.05
  4      ant             ""            6      0.001
  5      ant             ""            6      0.001
  6      ant             ''            6      0.001
  7      "queen ant"     'Ma\'am'      6      2e-3
  8      human           "Mark"        2      1.8
```

In this case it will identify the following columns:

```
Name       Type
----       ----
RECNO      Short
SPECIES    String
NAME       String
LEGS       Short
HEIGHT/m   Float
```

It will also use the text "`Here is a list of some animals`" as the Description parameter of the table. Without any of the comment lines, it would still interpret the table, but the columns would be given the names `col1`..`col5`.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`ascii(maxSample=5000)`". The following options are available:

`maxSample = <int>`
: Controls how many rows of the input file are sampled to determine column datatypes. When reading ASCII files, since no type information is present in the input file, the handler has to look at the column data to see what type of value appears to be present in each column, before even starting to read the data in. By default it goes through the whole table when doing this, which can be time-consuming for large tables. If this value is set, it limits the number of rows that are sampled in this data characterisation pass, which can reduce read time substantially. However, if values near the end of the table differ in apparent type from those near the start, it can also result in getting the datatypes wrong.

This format cannot be automatically identified by its content, so in general it is necessary to specify that a table is in ASCII format when reading it. However, if the input file has the extension "`.txt`" (case insensitive) an attempt will be made to read it using this format.

5.1.1.8 `ipac`

CalTech's Infrared Processing and Analysis Center use a text-based format for storage of tabular data, defined at http://irsa.ipac.caltech.edu/applications/DDGEN/Doc/ipac_tbl.html. Tables can

store column name, type, units and null values, as well as table parameters.

This format cannot be automatically identified by its content, so in general it is necessary to specify that a table is in IPAC format when reading it. However, if the input file has the extension "`.tbl`" or "`.ipac`" (case insensitive) an attempt will be made to read it using this format.

An example looks like this:

```
\Table name = "animals.vot"
\Description = "Some animals"
\Author = "Mark Taylor"
| RECNO | SPECIES   | NAME          | LEGS | HEIGHT | MAMMAL |
| int   | char      | char          | int  | double | char   |
|       |           |               |      | m      |        |
| null  | null      | null          | null | null   | null   |
  1       pig         Pigling Bland   4      0.8      true
  2       cow         Daisy           4      2.0      true
  3       goldfish    Dobbin          null   0.05     false
  4       ant         null            6      0.001    false
  5       ant         null            6      0.001    false
  6       queen ant   Ma'am           6      0.002    false
  7       human       Mark            2      1.8      true
```

5.1.1.9 `pds4`

NASA's Planetary Data System version 4 format is described at https://pds.nasa.gov/datastandards/. This implementation is based on v1.16.0 of PDS4.

PDS4 files consist of an XML *Label* file which provides detailed metadata, and which may also contain references to external data files stored alongside it. This input handler looks for (binary, character or delimited) tables in the Label; depending on the configuration it may restrict them to those in the `File_Area_Observational` area. The Label is the file which has to be presented to this input handler to read the table data. Because of the relationship between the label and the data files, it is usually necessary to move them around together.

If there are multiple tables in the label, you can refer to an individual one using the "`#`" specifier after the label file name by table `name`, `local_identifier`, or 1-based index (e.g. "`label.xml#1`" refers to the first table).

If there are `Special_Constants` defined in the label, they are in most cases interpreted as blank values in the output table data. At present, the following special values are interpreted as blanks: `saturated_constant`, `missing_constant`, `error_constant`, `invalid_constant`, `unknown_constant`, `not_applicable_constant`, `high_instrument_saturation`, `high_representation_saturation`, `low_instrument_saturation`, `low_representation_saturation`.

Fields within top-level Groups are interpreted as array values. Any fields in nested groups are ignored. For these array values only limited null-value substitution can be done (since array elements are primitives and so cannot take null values).

This input handler is somewhat experimental, and the author is not a PDS expert. If it behaves strangely or you have suggestions for how it could work better, please contact the author.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`pds4(checkmagic=false,observational=true)`". The following options are available:

`checkmagic = true|false`
Determines whether an initial test is made to see whether the file looks like PDS4 before

attempting to read it as one. The tests are ad-hoc and look for certain elements and namespaces that are expected to appear near the start of a table-containing PDS4 file, but it's not bulletproof. Setting this true is generally a good idea to avoid attempting to parse non-PDS4 files, but you can set it false to attempt to read an PDS4 file that starts with the wrong sequence.

`observational = true|false`
Determines whether only tables within a `<File_Area_Observational>` element of the PDS4 label should be included. If true, only observational tables are found, if false, other tables will be found as well.

Files in this format may contain multiple tables; depending on the context, either one or all tables will be read. Where only one table is required, either the first one in the file is used, or the required one can be specified after the "`#`" character at the end of the filename.

This format can be automatically identified by its content so you do not need to specify the format explicitly when reading PDS4 tables, regardless of the filename.

5.1.1.10 `mrt`

The so-called "Machine-Readable Table" format is used by AAS journals, and based on the format of readMe files used by the CDS. There is some documentation at https://journals.aas.org/mrt-standards/, which mostly builds on documentation at http://vizier.u-strasbg.fr/doc/catstd.htx, but the format is in fact quite poorly specified, so this input handler was largely developed on a best-efforts basis by looking at MRT tables actually in use by AAS, and with assistance from AAS staff. As such, it's not guaranteed to succeed in reading all MRT files out there, but it will try its best.

It only attempts to read MRT files themselves, there is currently no capability to read VizieR data tables which provide the header and formatted data in separate files; however, if a table is present in VizieR, there will be options to download it in more widely used formats that can be used instead.

An example looks like this:

```
Title: A search for multi-planet systems with TESS using a Bayesian
       N-body retrieval and machine learning
Author: Pearson K.A.
Table: Stellar Parameters
================================================================================
Byte-by-byte Description of file: ajab4e1ct2_mrt.txt
--------------------------------------------------------------------------------
   Bytes Format Units   Label   Explanations
--------------------------------------------------------------------------------
   1-  9 I9     ---     ID      TESS Input Catalog identifier
  11- 15 F5.2   mag     Tmag    Apparent TESS band magnitude
  17- 21 F5.3   solRad  R*      Stellar radius
  23- 26 I4     K       Teff    Effective temperature
  28- 32 F5.3   [cm/s2] log(g)  log surface gravity
  34- 38 F5.2   [Sun]   [Fe/H]  Metallicity
  40- 44 F5.3   ---     u1      Linear Limb Darkening
  46- 50 F5.3   ---     u2      Quadratic Limb Darkening
--------------------------------------------------------------------------------
231663901 12.35 0.860 5600 4.489  0.00 0.439 0.138
149603524  9.72 1.280 6280 4.321  0.24 0.409 0.140
336732616 11.46 1.400 6351 4.229  0.00 0.398 0.140
231670397  9.85 2.070 6036 3.934  0.00 0.438 0.117
...
```

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`mrt(checkmagic=false,errmode=FAIL)`". The following options are available:

checkmagic = true|false
Determines whether an initial test is made to see whether the file looks like MRT before attempting to read it as one; the test is that it starts with the string "`Title: `". Setting this true is generally a good idea to avoid attempting to parse non-MRT files, but you can set it false to attempt to read an MRT file that starts with the wrong sequence.

errmode = IGNORE|WARN|FAIL
Indicates what action should be taken if formatting errors are detected in the file at read time.

This format can be automatically identified by its content so you do not need to specify the format explicitly when reading MRT tables, regardless of the filename.

5.1.1.11 `parquet`

Parquet is a columnar format developed within the Apache project. Data is compressed on disk and read into memory before use.

This input handler will read columns representing scalars, strings and one-dimensional arrays of the same. It is not capable of reading multi-dimensional arrays, more complex nested data structures, or some more exotic data types like 96-bit integers. If such columns are encountered in an input file, a warning will be emitted through the logging system and the column will not appear in the read table. Support may be introduced for some additional types if there is demand.

At present, only very limited metadata is read. Parquet does not seem(?) to have any standard format for per-column metadata, so the only information read about each column apart from its datatype is its name.

Depending on the way that the table is accessed, the reader tries to take advantage of the column and row block structure of parquet files to read the data in parallel where possible.

Parquet support is currently somewhat experimental.

Note:

The parquet I/O handlers require large external libraries, which are not always bundled with the library/application software because of their size. In some configurations, parquet support may not be present, and attempts to read or write parquet files will result in a message like:

```
Parquet-mr libraries not available
```

If you can supply the relevant libaries on the classpath at runtime, the parquet support will work. At time of writing, the required libraries are included in the `topcat-extra.jar` monolithic jar file; they can also be found in the starjava github repository (https://github.com/Starlink/starjava/tree/master/parquet/src/lib, use `parquet-mr-stil.jar` and its dependencies), or you can acquire them from the Parquet MR package. These arrangements may be revised in future releases, for instance if parquet usage becomes more mainstream. The required dependencies are those of the Parquet MR submodule `parquet-cli`, in particular the files `parquet-cli-1.11.1.jar`, `parquet-column-1.11.1.jar`, `parquet-common-1.11.1.jar`, `parquet-encoding-1.11.1.jar`, `parquet-format-structures-1.11.1.jar`, `parquet-hadoop-1.11.1-noshaded.jar`, `parquet-jackson-1.11.1.jar`, `commons-collections-3.2.2.jar`, `commons-configuration-1.6.jar`, `commons-lang-2.6.jar`, `failureaccess-1.0.1.jar`, `guava-27.0.1-jre.jar`, `hadoop-auth-2.7.3.jar`, `hadoop-common-2.7.3.jar`, `log4j-1.2.17.jar`, `slf4j-api-1.7.22.jar`, `slf4j-log4j12-1.7.22.jar`, `snappy-java-1.1.7.3.jar`.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`parquet(cachecols=true,nThread=4)`". The following options are available:

`cachecols = true|false|null`
Forces whether to read all the column data at table load time. If `true`, then when the table is loaded, all data is read by column into local scratch disk files, which is generally the fastest way to ingest all the data. If `false`, the table rows are read as required, and possibly cached using the normal STIL mechanisms. If `null` (the default), the decision is taken automatically based on available information.

`nThread = <int>`
Sets the number of read threads used for concurrently reading table columns if the columns are cached at load time - see the `cachecols` option. If the value is <=0 (the default), a value is chosen based on the number of apparently available processors.

This format can be automatically identified by its content so you do not need to specify the format explicitly when reading parquet tables, regardless of the filename.

5.1.1.12 `feather`

The Feather file format is a column-oriented binary disk-based format based on Apache Arrow and supported by (at least) Python, R and Julia. Some description of it is available at https://github.com/wesm/feather and https://blog.rstudio.com/2016/03/29/feather/. It can be used for large datasets, but it does not support array-valued columns. It can be a useful format to use for exchanging data with R, for which FITS I/O is reported to be slow.

At present CATEGORY type columns are not supported, and metadata associated with TIME, DATE and TIMESTAMP columns is not retrieved.

This format can be automatically identified by its content so you do not need to specify the format explicitly when reading feather tables, regardless of the filename.

5.1.1.13 `gbin`

GBIN format is a special-interest file format used within DPAC, the Data Processing and Analysis Consortium working on data from the Gaia astrometry satellite. It is based on java serialization, and in all of its various forms has the peculiarity that you only stand any chance of decoding it if you have the Gaia data model classes on your java classpath at runtime. Since the set of relevant classes is very large, and also depends on what version of the data model your GBIN file corresponds to, those classes will not be packaged with this software, so some additional setup is required to read GBIN files.

As well as the data model classes, you must provide on the runtime classpath the GaiaTools classes required for GBIN reading. The table input handler accesses these by reflection, to avoid an additional large library dependency for a rather niche requirement. It is likely that since you have to supply the required data model classes you will also have the required GaiaTools classes to hand as well, so this shouldn't constitute much of an additional burden for usage.

In practice, if you have a jar file or files for pretty much any java library or application which is capable of reading a given GBIN file, just adding it or them to the classpath at runtime when using this input handler ought to do the trick. Examples of such jar files are the `MDBExplorerStandalone.jar` file available from https://gaia.esac.esa.int/mdbexp/, or the `gbcat.jar` file you can build from the CU9/software/gbcat/ directory in the DPAC subversion

repository.

The GBIN format doesn't really store tables, it stores arrays of java objects, so the input handler has to make some decisions about how to flatten these into table rows.

In its simplest form, the handler basically looks for public instance methods of the form `getXxx()` and uses the `Xxx` as column names. If the corresponding values are themselves objects with suitable getter methods, those objects are added as new columns instead. This more or less follows the practice of the `gbcat` (`gaia.cu1.tools.util.GbinInterogator`) tool. Method names are sorted alphabetically. Arrays of complex objects are not handled well, and various other things may trip it up. See the source code (e.g. `uk.ac.starlink.gbin.GbinTableProfile`) for more details.

If the object types stored in the GBIN file are known to the special metadata-bearing class `gaia.cu9.tools.documentationexport.MetadataReader` and its dependencies, and if that class is on the runtime classpath, then the handler will be able to extract additional metadata as available, including standardised column names, table and column descriptions, and UCDs. An example of a jar file containing this metadata class alongside data model classes is `GaiaDataLibs-18.3.1-r515078.jar`. Note however at time of writing there are some deficiencies with this metadata extraction functionality related to unresolved issues in the upstream gaia class libraries and the relevant interface control document (GAIA-C9-SP-UB-XL-034-01, "External Data Centres ICD"). Currently columns appear in the output table in a more or less random order, units and Utypes are not extracted, and using the GBIN reader tends to cause a 700kbyte file "temp.xml" to be written in the current directory. If the upstream issues are fixed, this behaviour may improve.

Note that support for GBIN files is somewhat experimental. Please contact the author (who is not a GBIN expert) if it doesn't seem to be working properly or you think it should do things differently.

This format can be automatically identified by its content so you do not need to specify the format explicitly when reading GBIN tables, regardless of the filename.

Example: Suppose you have the `MDBExplorerStandalone.jar` file containing the data model classes, you can read GBIN files by starting STILTS like this:

```
stilts -classpath MDBExplorerStandalone.jar ...
```

or like this:

```
java -classpath stilts.jar:MDBExplorerStandalone.jar uk.ac.starlink.ttools.Stilts ...
```

5.1.1.14 `tst`

Tab-Separated Table, or TST, is a text-based table format used by a number of astronomical tools including Starlink's GAIA and ESO's SkyCat on which it is based. A definition of the format can be found in Starlink Software Note 75. The implementation here ignores all comment lines: special comments such as the "`#column-units:`" are not processed.

An example looks like this:

```
Simple TST example; stellar photometry catalogue.

A.C. Davenhall (Edinburgh) 26/7/00.

Catalogue of U,B,V colours.
UBV photometry from Mount Pumpkin Observatory,
see Sage, Rosemary and Thyme (1988).

# Start of parameter definitions.
EQUINOX: J2000.0
EPOCH: J1996.35
```

```
id_col: -1
ra_col: 0
dec_col: 1

# End of parameter definitions.
ra<tab>dec<tab>V<tab>B_V<tab>U_B
--<tab>---<tab>-<tab>---<tab>---
5:09:08.7<tab> -8:45:15<tab>  4.27<tab>  -0.19<tab>  -0.90
5:07:50.9<tab> -5:05:11<tab>  2.79<tab>  +0.13<tab>  +0.10
5:01:26.3<tab> -7:10:26<tab>  4.81<tab>  -0.19<tab>  -0.74
5:17:36.3<tab> -6:50:40<tab>  3.60<tab>  -0.11<tab>  -0.47
[EOD]
```

This format cannot be automatically identified by its content, so in general it is necessary to specify that a table is in TST format when reading it.

5.1.1.15 `wdc`

Some support is provided for files produced by the World Data Centre for Solar Terrestrial Physics. The format itself apparently has no name, but files in this format look something like the following:

```
Column formats and units - (Fixed format columns which are single space separated.)
------------------------
Datetime (YYYY mm dd HHMMSS)            %4d %2d %2d %6d      -
                                        %1s
aa index - 3-HOURLY (Provisional)       %3d                  nT

2000 01 01 000000  67
2000 01 01 030000  32
    ...
```

Support for this (obsolete?) format may not be very complete or robust.

This format cannot be automatically identified by its content, so in general it is necessary to specify that a table is in WDC format when reading it.

5.1.2 Output Formats

Some of the tools ask you to specify the format of output tables using the `ofmt` parameter. The output formats described below are supported; in some cases there are variants or options for the basic formats as documented. If you don't specify an output format explicitly, STILTS will try to guess what format to write based on the output filename; the details of those rules are also documented below.

It is also possible to add new formats at runtime using the `startable.writers` system property, or by setting the format to the classname of a `uk.ac.starlink.table.StarTableWriter` class.

5.1.2.1 `fits`

FITS is a very well-established format for storage of astronomical table or image data (see https://fits.gsfc.nasa.gov/). This writer stores tables in a FITS file consisting of two HDUs (Header+Data Units): a Primary HDU as required by the FITS standard, and a single extension of type BINTABLE containing the table data.

There are a few variants of this format:

fits-plus
: The primary HDU contains an array of bytes which stores the full table metadata as the text of a VOTable document, along with headers that mark this has been done. Most FITS table readers will ignore this altogether and treat the file just as if it contained only the table. When

it is re-read by this or compatible applications however, they can read out the metadata and make it available for use. In this way you can store your data in the efficient and widely portable FITS format without losing the additional metadata such as table parameters, column UCDs, lengthy column descriptions etc that may be attached to the table.

fits-basic
The primary HDU contains only very minimal headers and no data.

fits-var
Behaves like `fits-basic`, but columns containing variable-length numeric array data are stored using the `P` and `Q` formats where appropriate, rather than padding smaller arrays to the size of the largest. This can make for more compact storage of variable-length array-valued column data but may also result in tables less suitable for streaming.

fits-healpix
Used for storing HEALPix pixel data in a way that conforms to the HEALPix-FITS serialization convention. In most ways it behaves the same as `fits-basic`, but it will rearrange and rename columns as required to follow the convention, and it will fail if the table does not contain the required HEALPix metadata (`STIL_HPX_*` parameters).

The default output format is `fits-plus`; in general you don't need to worry about this, it just gives you some hidden benefits over `fits-basic`.

A private convention is used where required to support encoding of tables with more than 999 columns (not possible in standard FITS); this was discussed on the FITSBITS mailing list in July 2017 in the thread BINTABLE convention for >999 columns. If software unaware of this convention (e.g. CFITSIO) is used to read such tables, it will only see the first 998 columns written as intended, plus a column 999 containing an undescribed byte buffer.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`fits-plus(date=true)`". The following options are available:

`date = true|false`
If true, the DATE-HDU header is filled in with the current date; otherwise it is not included.

Multiple tables may be written to a single output file using this format.

If no output format is explicitly chosen, writing to a filename with the extension "`.fit`", "`.fits`" or "`.fts`" (case insensitive) will select `fits-plus` format for output.

For column-oriented FITS table output, see Section 5.1.2.2.

5.1.2.2 `colfits`

Column-oriented FITS output consists of a FITS file containing two HDUs (Header+Data Units); a primary one (required by the FITS standard) and a single extension of type BINTABLE containing the table data. Unlike normal FITS format however, this table consists of a single row in which each cell holds the data for an entire column.

This can be a more efficient format to work with when dealing with very large, and especially very wide, tables. The benefits are greatest when the file size exceeds the amount of available physical memory and operations are required which scan through the table using only a few of the columns (many common operations, for instance plotting two columns against each other, fall into this category). The overhead for reading and writing this format is somewhat higher than for normal FITS however, and other applications may not be able to work with it (though it is a legal FITS file), so in most cases normal FITS is a more suitable choice.

There are two variants of this format:

colfits-plus
The primary HDU contains an array of bytes which stores the table metadata in VOTable format.

colfits-basic
The primary HDU contains no data.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`colfits-plus(date=true)`". The following options are available:

`date = true|false`
If true, the DATE-HDU header is filled in with the current date; otherwise it is not included.

Multiple tables may be written to a single output file using this format.

If no output format is explicitly chosen, writing to a filename with the extension "`.colfits`" (case insensitive) will select `colfits-plus` format for output.

For normal (row-oriented) FITS output, see Section 5.1.2.1.

5.1.2.3 `votable`

VOTable is an XML-based format for tabular data endorsed by the International Virtual Observatory Alliance and defined in the VOTable Recommendation. While the tabular data which can be encoded is by design close to what FITS allows, it provides for much richer encoding of structure and metadata. Most of the table data exchanged by VO services is in VOTable format, but it can be used for local table storage as well.

When a table is saved to VOTable format, a document conforming to the VOTable specification containing a single TABLE element within a single RESOURCE element is written. Where the table contains such information (often obtained by reading an input VOTable), column and table metadata will be written out as appropriate to the attributes `unit`, `ucd`, `xtype` and `utype`, and the elements `COOSYS`, `TIMESYS` and `DESCRIPTION` attached to table columns or parameters.

There are various ways that a VOTable can be written; by default the output serialization format is TABLEDATA and the VOTable format version is 1.4, or a value controlled by the `votable.version` system property. However, configuration options are available to adjust these defaults.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`votable(format=BINARY2,version=V13)`". The following options are available:

`format = TABLEDATA|BINARY|BINARY2|FITS`
Gives the serialization type (DATA element content) of output VOTables.

`version = V10|V11|V12|V13|V14`
Gives the version of the VOTable format which will be used when writing the VOTable. "`V10`" is version 1.0 etc.

`inline = true|false`
If true, STREAM elements are written base64-encoded within the body of the document, and if false they are written to a new external binary file whose name is derived from that of the output VOTable document. This is only applicable to BINARY, BINARY2 and FITS formats

where output is not to a stream.

compact = true|false|null
Controls whitespace formatting for TABLEDATA output, ignored for other formats. By default a decision will be taken dependent on table width.

encoding = UTF-8|UTF-16|...
Specifies the XML encoding used in the output VOTable. The default value is UTF-8. Note that certain optimisations are in place for UTF-8 output which means that other encodings may be significantly slower.

Multiple tables may be written to a single output file using this format.

If no output format is explicitly chosen, writing to a filename with the extension ".vot", ".votable" or ".xml" (case insensitive) will select votable format for output.

5.1.2.4 csv

Writes tables in the semi-standard Comma-Separated Values format. This does not preserve any metadata apart from column names, and is generally inefficient to read, but it can be useful for importing into certain external applications, such as some databases or spreadsheets.

By default, the first line is a header line giving the column names, but this can be inhibited using the header=false configuration option.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "csv(header=true,maxCell=160)". The following options are available:

header = true|false
If true, the first line of the CSV output will be a header containing the column names; if false, no header line is written and all lines represent data rows.

maxCell = <int>
Maximum width in characters of an output table cell. Cells longer than this will be truncated.

If no output format is explicitly chosen, writing to a filename with the extension ".csv" (case insensitive) will select CSV format for output.

An example looks like this:

```
RECNO,SPECIES,NAME,LEGS,HEIGHT,MAMMAL
1,pig,Pigling Bland,4,0.8,true
2,cow,Daisy,4,2.0,true
3,goldfish,Dobbin,,0.05,false
4,ant,,6,0.001,false
5,ant,,6,0.001,false
6,queen ant,Ma'am,6,0.002,false
7,human,Mark,2,1.8,true
```

5.1.2.5 ecsv

The Enhanced Character Separated Values format was developed within the Astropy project and is described in Astropy APE6 (DOI). It is composed of a YAML header followed by a CSV-like body, and is intended to be a human-readable and maybe even human-writable format with rich metadata. Most of the useful per-column and per-table metadata is preserved when de/serializing to this format. The version supported by this writer is currently ECSV 1.0.

ECSV allows either a space or a comma for delimiting values, controlled by the `delimiter` configuration option. If `ecsv(delimiter=comma)` is used, then removing the YAML header will leave a CSV file that can be interpreted by the CSV inputhandler or imported into other CSV-capable applications.

Following the ECSV 1.0 specification, array-valued columns are supported. ECSV 1.0, required for working with array-valued columns, is supported by Astropy v4.3 and later.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`ecsv(delimiter=comma)`". The following options are available:

`delimiter = comma|space`
Delimiter character, which for ECSV may be either a space or a comma. Permitted values are "`space`" or "`comma`".

If no output format is explicitly chosen, writing to a filename with the extension "`.ecsv`" (case insensitive) will select `ECSV` format for output.

An example looks like this:

```
# %ECSV 1.0
# ---
# datatype:
# -
#   name: RECNO
#   datatype: int32
# -
#   name: SPECIES
#   datatype: string
# -
#   name: NAME
#   datatype: string
#   description: How one should address the animal in public & private.
# -
#   name: LEGS
#   datatype: int32
#   meta:
#     utype: anatomy:limb
# -
#   name: HEIGHT
#   datatype: float64
#   unit: m
#   meta:
#     VOTable precision: 2
# -
#   name: MAMMAL
#   datatype: bool
# meta:
#   name: animals.vot
#   Description: Some animals
#   Author: Mark Taylor
RECNO SPECIES NAME LEGS HEIGHT MAMMAL
1 pig "Pigling Bland" 4 0.8 True
2 cow Daisy 4 2.0 True
3 goldfish Dobbin "" 0.05 False
4 ant "" 6 0.001 False
5 ant "" 6 0.001 False
6 "queen ant" Ma'am 6 0.002 False
7 human Mark 2 1.8 True
```

5.1.2.6 `ascii`

Writes to a simple plain-text format intended to be comprehensible by humans or machines.

The first line is a comment, starting with a "#" character, naming the columns, and an attempt is made to line up data in columns using spaces. No metadata apart from column names is written.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`ascii(maxCell=158,maxParam=160)`". The following options are available:

`maxCell = <int>`
Maximum width in characters of an output table cell. Cells longer than this will be truncated.

`maxParam = <int>`
Maximum width in characters of an output table parameter. Parameters with values longer than this will be truncated.

`params = true|false`
Whether to output table parameters as well as row data.

If no output format is explicitly chosen, writing to a filename with the extension "`.txt`" (case insensitive) will select `ascii` format for output.

An example looks like this:

```
# RECNO SPECIES   NAME          LEGS HEIGHT MAMMAL
  1     pig       "Pigling Bland" 4    0.8    true
  2     cow       Daisy         4    2.0    true
  3     goldfish  Dobbin        ""   0.05   false
  4     ant       ""            6    0.001  false
  5     ant       ""            6    0.001  false
  6     "queen ant" "Ma\'am"      6    0.002  false
  7     human     Mark          2    1.8    true
```

5.1.2.7 `ipac`

Writes output in the format used by CalTech's Infrared Processing and Analysis Center, and defined at http://irsa.ipac.caltech.edu/applications/DDGEN/Doc/ipac_tbl.html. Column name, type, units and null values are written, as well as table parameters.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`ipac(maxCell=1000,maxParam=100000)`". The following options are available:

`maxCell = <int>`
Maximum width in characters of an output table cell. Cells longer than this will be truncated.

`maxParam = <int>`
Maximum width in characters of an output table parameter. Parameters with values longer than this will be truncated.

`params = true|false`
Whether to output table parameters as well as row data.

If no output format is explicitly chosen, writing to a filename with the extension "`.tbl`" or "`.ipac`" (case insensitive) will select `IPAC` format for output.

An example looks like this:

```
\Table name = "animals.vot"
\Description = "Some animals"
\Author = "Mark Taylor"
```

```
| RECNO | SPECIES   | NAME          | LEGS | HEIGHT | MAMMAL |
| int   | char      | char          | int  | double | char   |
|       |           |               |      | m      |        |
| null  | null      | null          | null | null   | null   |
  1       pig         Pigling Bland   4      0.8      true
  2       cow         Daisy           4      2.0      true
  3       goldfish    Dobbin          null   0.05     false
  4       ant         null            6      0.001    false
  5       ant         null            6      0.001    false
  6       queen ant   Ma'am           6      0.002    false
  7       human       Mark            2      1.8      true
```

5.1.2.8 `parquet`

Parquet is a columnar format developed within the Apache project. Data is compressed on disk and read into memory before use.

At present, only very limited metadata is written. Parquet does not seem(?) to have any standard format for per-column metadata, so the only information written about each column apart from its datatype is its name.

Parquet support is currently somewhat experimental.

Note:

The parquet I/O handlers require large external libraries, which are not always bundled with the library/application software because of their size. In some configurations, parquet support may not be present, and attempts to read or write parquet files will result in a message like:

```
Parquet-mr libraries not available
```

If you can supply the relevant libaries on the classpath at runtime, the parquet support will work. At time of writing, the required libraries are included in the `topcat-extra.jar` monolithic jar file; they can also be found in the starjava github repository (https://github.com/Starlink/starjava/tree/master/parquet/src/lib, use `parquet-mr-stil.jar` and its dependencies), or you can acquire them from the Parquet MR package. These arrangements may be revised in future releases, for instance if parquet usage becomes more mainstream. The required dependencies are those of the Parquet MR submodule `parquet-cli`, in particular the files `parquet-cli-1.11.1.jar`, `parquet-column-1.11.1.jar`, `parquet-common-1.11.1.jar`, `parquet-encoding-1.11.1.jar`, `parquet-format-structures-1.11.1.jar`, `parquet-hadoop-1.11.1-noshaded.jar`, `parquet-jackson-1.11.1.jar`, `commons-collections-3.2.2.jar`, `commons-configuration-1.6.jar`, `commons-lang-2.6.jar`, `failureaccess-1.0.1.jar`, `guava-27.0.1-jre.jar`, `hadoop-auth-2.7.3.jar`, `hadoop-common-2.7.3.jar`, `log4j-1.2.17.jar`, `slf4j-api-1.7.22.jar`, `slf4j-log4j12-1.7.22.jar`, `snappy-java-1.1.7.3.jar`.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`parquet(groupArray=false)`". The following options are available:

`groupArray = true|false`
Controls the low-level detail of how array-valued columns are written. For an array-valued int32 column named IVAL, `groupArray=false` will write it as "`repeated int32 IVAL`" while `groupArray=true` will write it as "`optional group IVAL (LIST) { repeated group list { optional int32 item} }`". I don't know why you'd want to do it the latter way, but some other parquet writers seem to do that by default, so there must be some good reason.

If no output format is explicitly chosen, writing to a filename with the extension ".parquet" or ".parq" (case insensitive) will select parquet format for output.

5.1.2.9 feather

The Feather file format is a column-oriented binary disk-based format based on Apache Arrow and supported by (at least) Python, R and Julia. Some description of it is available at https://github.com/wesm/feather and https://blog.rstudio.com/2016/03/29/feather/. It can be used for large datasets, but it does not support array-valued columns. It can be a useful format to use for exchanging data with R, for which FITS I/O is reported to be slow.

This writer is somewhat experimental; please report problems if you encounter them.

If no output format is explicitly chosen, writing to a filename with the extension ".fea" or ".feather" (case insensitive) will select feather format for output.

5.1.2.10 text

Writes tables in a simple text-based format designed to be read by humans. No reader exists for this format.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "text(maxCell=40,maxParam=160)". The following options are available:

maxCell = <int>
Maximum width in characters of an output table cell. Cells longer than this will be truncated.

maxParam = <int>
Maximum width in characters of an output table parameter. Parameters with values longer than this will be truncated.

params = true|false
Whether to output table parameters as well as row data.

Multiple tables may be written to a single output file using this format.

An example looks like this:

```
Table name: animals.vot
Description: Some animals
Author: Mark Taylor
+-------+-----------+---------------+------+--------+--------+
| RECNO | SPECIES   | NAME          | LEGS | HEIGHT | MAMMAL |
+-------+-----------+---------------+------+--------+--------+
| 1     | pig       | Pigling Bland | 4    | 0.8    | true   |
| 2     | cow       | Daisy         | 4    | 2.0    | true   |
| 3     | goldfish  | Dobbin        |      | 0.05   | false  |
| 4     | ant       |               | 6    | 0.001  | false  |
| 5     | ant       |               | 6    | 0.001  | false  |
| 6     | queen ant | Ma'am         | 6    | 0.002  | false  |
| 7     | human     | Mark          | 2    | 1.8    | true   |
+-------+-----------+---------------+------+--------+--------+
```

5.1.2.11 html

Writes a basic HTML TABLE element suitable for use as a web page or for insertion into one.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`html(maxCell=200,standalone=false)`". The following options are available:

`maxCell = <int>`
Maximum width in characters of an output table cell. Cells longer than this will be truncated.

`standalone = true|false`
If true, the output is a freestanding HTML document complete with HTML, HEAD and BODY tags. If false, the output is just a TABLE element.

Multiple tables may be written to a single output file using this format.

If no output format is explicitly chosen, writing to a filename with the extension "`.html`" or "`.htm`" (case insensitive) will select `HTML` format for output.

An example looks like this:

```
<TABLE BORDER='1'>
<CAPTION><STRONG>animals.vot</STRONG></CAPTION>
<THEAD>
<TR> <TH>RECNO</TH> <TH>SPECIES</TH> <TH>NAME</TH> <TH>LEGS</TH> <TH>HEIGHT</TH> <TH>MAMMAL</T
<TR> <TH> </TH> <TH> </TH> <TH> </TH> <TH> </TH> <TH>(m)</TH> <TH> </
<TR><TD colspan='6'></TD></TR>
</THEAD>
<TBODY>
<TR> <TD>1</TD> <TD>pig</TD> <TD>Pigling Bland</TD> <TD>4</TD> <TD>0.8</TD> <TD>true</TD></TR>
<TR> <TD>2</TD> <TD>cow</TD> <TD>Daisy</TD> <TD>4</TD> <TD>2.0</TD> <TD>true</TD></TR>
<TR> <TD>3</TD> <TD>goldfish</TD> <TD>Dobbin</TD> <TD> </TD> <TD>0.05</TD> <TD>false</TD>
<TR> <TD>4</TD> <TD>ant</TD> <TD> </TD> <TD>6</TD> <TD>0.001</TD> <TD>false</TD></TR>
<TR> <TD>5</TD> <TD>ant</TD> <TD> </TD> <TD>6</TD> <TD>0.001</TD> <TD>false</TD></TR>
<TR> <TD>6</TD> <TD>queen ant</TD> <TD>Ma&apos;am</TD> <TD>6</TD> <TD>0.002</TD> <TD>false</TD
<TR> <TD>7</TD> <TD>human</TD> <TD>Mark</TD> <TD>2</TD> <TD>1.8</TD> <TD>true</TD></TR>
</TBODY>
</TABLE>
```

5.1.2.12 `latex`

Writes a table as a LaTeX `tabular` environment, suitable for insertion into a document intended for publication. This is only likely to be useful for fairly small tables.

The handler behaviour may be modified by specifying one or more comma-separated name=value configuration options in parentheses after the handler name, e.g. "`latex(standalone=false)`". The following options are available:

`standalone = true|false`
If true, the output is a freestanding LaTeX document consisting of a `tabular` environment within a `table` within a `document`. If false, the output is just a `tabular` environment.

If no output format is explicitly chosen, writing to a filename with the extension "`.tex`" (case insensitive) will select `LaTeX` format for output.

An example looks like this:

```
\begin{tabular}{|r|l|l|r|r|l|}
\hline
  \multicolumn{1}{|c|}{RECNO} &
  \multicolumn{1}{c|}{SPECIES} &
  \multicolumn{1}{c|}{NAME} &
  \multicolumn{1}{c|}{LEGS} &
  \multicolumn{1}{c|}{HEIGHT} &
```

```
  \multicolumn{1}{c|}{MAMMAL} \\
\hline
  1 & pig & Pigling Bland & 4 & 0.8 & true\\
  2 & cow & Daisy & 4 & 2.0 & true\\
  3 & goldfish & Dobbin &  & 0.05 & false\\
  4 & ant &  & 6 & 0.001 & false\\
  5 & ant &  & 6 & 0.001 & false\\
  6 & queen ant & Ma'am & 6 & 0.002 & false\\
  7 & human & Mark & 2 & 1.8 & true\\
\hline\end{tabular}
```

5.1.2.13 `tst`

Tab-Separated Table, or TST, is a text-based table format used by a number of astronomical tools including Starlink's GAIA and ESO's SkyCat on which it is based. A definition of the format can be found in Starlink Software Note 75.

If no output format is explicitly chosen, writing to a filename with the extension "`.tst`" (case insensitive) will select `TST` format for output.

An example looks like this:

```
animals.vot

# Table parameters
Description: Some animals
Author: Mark Taylor

# Attempted guesses about identity of columns in the table.
# These have been inferred from column UCDs and/or names
# in the original table data.
# The algorithm which identifies these columns is not particularly reliable,
# so it is possible that these are incorrect.
id_col: 2
ra_col: -1
dec_col: -1

# This TST file generated by STIL v4.1-2

RECNO SPECIES NAME LEGS HEIGHT MAMMAL
----- ------- ---- ---- ------ ------
1 pig Pigling Bland 4 0.8 true
2 cow Daisy 4 2.0 true
3 goldfish Dobbin         0.05 false
4 ant         6 0.001 false
5 ant         6 0.001 false
6 queen ant Ma'am 6 0.002 false
7 human Mark 2 1.8 true
[EOD]
```

5.1.2.14 `mirage`

Mirage was a nice standalone tool for analysis of multidimensional data, from which TOPCAT took some inspiration. It was described in a 2007 paper 2007ASPC..371..391H, but no significant development seems to have taken place since then. This format is therefore probably obsolete, but you can still write table output in Mirage-compatible format if you like.

If no output format is explicitly chosen, writing to a filename with the extension "`.mirage`" (case insensitive) will select `mirage` format for output.

An example looks like this:

```
#
# Written by uk.ac.starlink.mirage.MirageFormatter
# Omitted column 5: MAMMAL(Boolean)
```

```
#
# Column names
format var RECNO SPECIES NAME LEGS HEIGHT
#
# Text columns
format text SPECIES
format text NAME
#
# Table data
1 pig Pigling_Bland 4 0.8
2 cow Daisy 4 2.0
3 goldfish Dobbin <blank> 0.05
4 ant <blank> 6 0.001
5 ant <blank> 6 0.001
6 queen_ant Ma'am 6 0.002
7 human Mark 2 1.8
```

5.2 Input Locations

The location of a serialized input table, usually given using the `in` parameter or similar, may be given in one of the forms listed below.

Filename
Very often, you will simply specify a filename as location, and the tool will just read from it in the usual way.

URL
Tables can be read from URLs directly. Some non-standard URL protocols are supported as well as the usual ones. The list is:

`http:`
Read from HTTP resources.

`https:`
Read from HTTPS resources.

`ftp:`
Read from anonymous FTP resources.

`file:`
Read from local files, using the syntax `file:///path/to/file`. This is similar to specifying the filename directly, but there is a difference: using this form forces reads to be sequential rather than random access, which may allow you to experience a different set of performance characteristics and bugs.

`jar:`
Specialised protocol for looking inside Java Archive files - see JarURLConnection documentation.

`myspace:`
(Obsolete?) Accesses files in the AstroGrid "MySpace" virtual file store. These URLs look something like "`myspace:/survey/iras_psc.xml`", and can access files in the myspace are that the user is currently logged into. These URLs can be used for both input and output of tables. To use them you must have an AstroGrid account and the AstroGrid WorkBench or similar must be running; if you're not currently logged in a dialogue will pop up to ask you for name and password.

`ivo:`
(Obsolete?) Understands ivo-type URLs which signify files in the AstroGrid "MySpace" virtual file store. These URLs look something like "`ivo://uk.ac.le.star/filemanager#node-2583`". These URLs can be used for both input and output of tables. To use them you must have an AstroGrid account and the AstroGrid WorkBench or similar must be running; if you're not currently logged in a

dialogue will pop up to ask you for name and password.

`jdbc:`
JDBC URLs may be used, but they don't work in the same way as the others listed here, since they do not reference an input byte stream. See instead Section 5.3.3.

Minus sign ("-")
The special location "-" (minus sign) indicates standard input. This allows you to use STILTS commands in a normal Unix pipeline.

System command ("< *syscmd*" or "*syscmd* |")
If the location starts with a "<" character or ends with a "|" character, the rest of the string is taken as a command line to be executed by the system shell. For instance a location like "`<cat header.txt data.txt`" (or equivalently "`cat header.txt data.txt|`") could be used to prepend a header line to an ASCII data file before it is passed to the STILTS ASCII-format input handler. Note this syntax will probably only work on Unix-like systems.

In any of these cases, for input locations compression is taken care of automatically. That means that you can give the filename or URL of a file which is compressed using `gzip`, `bzip2` or Unix `compress` and the program will uncompress it on the fly.

For file formats that can contain multiple tables, the one required, if it's not the first in the file, can generally be specified by a position indicator string appended to the basic location following a "#". For instance "cat.fits#3" references HDU 3 in a multi-extension FITS file. The details of this syntax depend on the file format, and are given in the relevant subsection of Section 5.1.1.

Note that tables can also be supplied from non-serialized sources, as described in Input Schemes.

5.3 Input Schemes

As well as being able to load tables from external data streams, STILTS offers a way to specify tables that do not correspond to a stream of bytes. These may be defined programmatically or interact with external services in some way that is not as straightforward as decoding a stream of bytes.

Such tables are defined using different *schemes*, and scheme specifications may be used in the same places as input table names, for instance as the value of the `in` parameter in `tpipe` and other commands. If an input location parameter (`in`) uses a scheme specification, the corresponding input format parameter (`ifmt`) is ignored.

The form of a scheme specification is:

```
:<scheme-name>:<scheme-specific-part>
```

so that for instance "`in=:loop:10`" specifies a 10-row single-column table, as described by the loop scheme documentation below. For an example of using such tables, you can try running for instance

```
stilts plot2plane in=:attractor:1e7,clifford layer1=mark shading1=density densemap1=plasma
```

The following subsections describe all the schemes that are available by default. It is also possible to add new schemes at runtime by using the `startable.schemes` system property.

5.3.1 `skysim`

Usage: `:skysim:<nrow>`

Generates a simulated all-sky star catalogue with a specified number of rows. This is intended to provide crude test catalogues when no suitable real dataset of the required size is available. In the current implementation the row count, which may be given in integer or exponential notation, is the only parameter, so the specification "`:skysim:5e6`" would give a 5 million-row simulated catalogue.

The current implementation provides somewhat realistic position-dependent star densities and distributions of magnitudes and colours based on positionally averaged values from Gaia EDR3. The source positions do not correspond to actual stars. The columns and the statistics on which the output is based may change in future releases.

Example:

```
:skysim:6
+-----------+------------+------------+------------+-----------+-----------+------------+
| ra        | dec        | l          | b          | gmag      | rmag      | b_r        |
+-----------+------------+------------+------------+-----------+-----------+------------+
| 266.7702  | -32.689117 | -3.044958  | -2.217145  | 18.168278 | 16.03555  | 1.046624   |
| 276.83398 | 8.132022   | 37.491447  | 9.031909   | 18.331224 | 18.786451 | 1.4425725  |
| 92.04118  | 23.79857   | -173.02219 | 1.7854756  | 16.743847 | 15.623316 | 1.690048   |
| 271.08215 | -5.2012086 | 22.848532  | 8.022958   | 21.538874 | 17.782997 | 2.1645386  |
| 298.83368 | 31.401922  | 67.75605   | 1.6348709  | 20.145718 | 17.728764 | 1.1521724  |
| 204.9299  | -77.07571  | -54.29949  | -14.464215 | 19.044079 | 20.277771 | 0.92987275 |
+-----------+------------+------------+------------+-----------+-----------+------------+
```

5.3.2 `attractor`

Usage: `:attractor:<nrow>[,(clifford[,a,b,c,d]|rampe[,a,b,c,d,e,f]|henon[,a,b,c])]`

Generates tables listing points sampled from one of a specified family of strange attractors. These can provide tables with (X,Y) or (X,Y,Z) columns and arbitrarily many rows. They can be used, for instance, to make (beautiful) example large-scale scatter plots in 2-d or 3-d space.

The specification syntax is of the form `:attractor:<nrow>,<family-name>[,<args>]` where `<nrow>` is the number of rows required, `<family-name>` is the name of one of the supported families of attractors, and `<args>` is an optional comma-separated list of numeric arguments specifying the family-specific parameters of the required attractor. If the `<args>` part is omitted, an example attractor from the family is used. Note that picking `<args>` values at random will often result in rather boring (non-strange) attractors.

The following families are currently supported:

clifford
: clifford attractors are 2-dimensional and have 4 parameters, with suggested values in the range +/-2.0.

  The iteration is defined by the equations:

```
x' = sin(a*y) + c * cos(a*x)
y' = sin(b*x) + d * cos(b*y)
```

  Examples:

- `:attractor:9999,clifford`
- `:attractor:1e6,clifford,1.32,-1.44,-1.7,-1.58`
- `:attractor:65536,clifford,1.27,-1.35,0.82,1.8`
- `:attractor:1e7,clifford,-1.9,1.18,-1.21,1.07`
- `:attractor:400,clifford,-1.27,-1.28,1.0,-1.26`
- `:attractor:4e6,clifford,1.8,0.9,-1.8,0.8`

rampe

rampe attractors are 3-dimensional and have 6 parameters, with suggested values in the range +/-2.0.

The iteration is defined by the equations:

```
x' = x * z * sin(a*x) - cos(b*y)
y' = y * x * sin(c*y) - cos(d*z)
z' = z * y * sin(e*z) - cos(f*x)
```

Examples:

- `:attractor:10e6,rampe`
- `:attractor:4,rampe,-1.81,1.35,-0.85,0.32,1.68,-1.62`
- `:attractor:5.5e5,rampe,0.23,-1.77,1.32,-1.44,-1.7,-1.58`
- `:attractor:9999,rampe,-0.3,1.78,-0.87,1.69,1.42,1.21`
- `:attractor:1e6,rampe,1.42,-1.98,0.39,1.32,1.79,-0.37`

henon

henon attractors are 2-dimensional and have 3 parameters, with suggested values in the range +/-2.0.

The iteration is defined by the equations:

```
x' = y + a + b*x*x
y' = c*x
```

Examples:

- `:attractor:65536,henon`
- `:attractor:1e7,henon,-0.68,1.64,0.36`
- `:attractor:400,henon,1.73,0.29,-0.99`
- `:attractor:4e6,henon,0.88,-0.9,-0.93`
- `:attractor:10e6,henon,1.4,-1.13,-0.01`

Example:

```
:attractor:6,rampe
+----------------------+---------------------+----------------------+
| x                    | y                   | z                    |
+----------------------+---------------------+----------------------+
| -0.5759098296568739  | 0.09844750286352466 | -0.6712534741282851  |
| -1.3295344852011892  | -0.9829776649068059 | -0.7814409891660122  |
| -1.1910376215054008  | 0.04335596646295736 | -1.0308958690758545  |
| -2.0144704755218514  | -0.9699626185329038 | -0.35169532148364757 |
| -0.16145296509226564 | 0.5245428249077974  | 0.17929370340580017  |
| -0.8409807675257591  | -0.9598486078341374 | -0.955769158222801   |
+----------------------+---------------------+----------------------+
```

5.3.3 `jdbc`

Usage: `:jdbc:<jdbc-part>`

Interacts with the JDBC system (JDBC sort-of stands for Java DataBase Connectivity) to execute an SQL query on a connected database. The `jdbc:...` specification is the JDBC URL. For historical compatibility reasons, specifications of this scheme may omit the leading colon character, so that the following are both legal, and are equivalent:

```
jdbc:mysql://localhost/dbl#SELECT TOP 10 ra, dec FROM gsc
:jdbc:mysql://localhost/dbl#SELECT TOP 10 ra, dec FROM gsc
```

In order for this to work, you must have access to a suitable database with a JDBC driver, and some standard JDBC configuration is required to set the driver up. The following steps are necessary:

1. the driver class must be available on the runtime classpath
2. the `jdbc.drivers` system property must be set to the driver classname

More detailed information about how to set up the JDBC system to connect with an available database, and of how to construct JDBC URLs, is provided elsewhere in the documentation.

5.3.4 `loop`

Usage: `:loop:<count>|<start>,<end>[,<step>]`

Generates a table whose single column increments over a given range.

The specification may either be a single value N giving the number of rows, which yields values in the range 0..N-1, or two or three comma-separated values giving the *start*, *end* and optionally *step* corresponding to the conventional specification of a loop variable.

The supplied numeric parameters are interpreted as floating point values, but the output column type will be 32- or 64-bit integer or 64-bit floating point, depending on the values that it has to take.

Examples:

- `:loop:5`: a 5-row table whose integer column has values 0, 1, 2, 3, 4
- `:loop:10,20`: a 10-row table whose integer column has values 10, 11, ... 19
- `:loop:1,2,0.25`: a 10-row table whose floating point column has values 1.00, 1.25, 1.50, 1.75
- `:loop:1e10`: a ten billion row table, with 64-bit integer values

Example:

```
:loop:6
+---+
| i |
+---+
| 0 |
| 1 |
| 2 |
| 3 |
| 4 |
| 5 |
+---+
```

5.3.5 `test`

Usage: `:test:[<nrow>[,<opts-ibsfvmk*>]]`

Generates a table containing test data. The idea is to include columns of different data types, for instance to provide an example for testing I/O handler implementations. The columns will contain some variety of more or less meaningless values, but the content is reproducible between runs, so the same specification will produce the same output each time. Updates of the implementation might change the output however, so the output is not guaranteed to be the same for all time.

The table specification has two comma-separated parameters:

- `<nrow>`: row count
- `<opts>`: a string of letter options specifying what types of data will be included; options are:

- **i**: an integer index column
- **b**: a few basic columns
- **s**: a selection of typed scalar columns
- **f**: a selection of fixed-length 1-d array columns
- **v**: a selection of variable-length 1-d array columns
- **m**: a selection of multi-dimensional array columns
- **k**: almost a thousand columns
- *****: equivalent to all of the above

If `<opts>` and/or `<nrow>` are omitted, some default values are used.

Example:

```
:test:10,is
+---------+--------+---------+-------+--------+---------+----------+----------------+---------
| i_index | s_byte | s_short | s_int | s_long | s_float | s_double | s_string       | s_boolea
+---------+--------+---------+-------+--------+---------+----------+----------------+---------
| 0       | 0      | 0       | 0     | 0      | 0.0     | 0.0      | zero           | false
| 1       |        | 1       | 1     | 1      | 1.0     | 1.0      | one            | true
| 2       | 2      |         | 2     | 2      | 2.0     | 2.0      | two            | false
| 3       | 3      | 3       |       | 3      | 3.0     | 3.0      | three          | true
| 4       | 4      | 4       | 4     |        | 4.0     | 4.0      | four           | false
| 5       | 5      | 5       | 5     | 5      |         | 5.0      | five           | true
| 6       | 6      | 6       | 6     | 6      | 6.0     |          | six            | false
| 7       | 7      | 7       | 7     | 7      | 7.0     | 7.0      |                | true
| 8       | 8      | 8       | 8     | 8      | 8.0     | 8.0      | ' "\""' ; '&<> |
| 9       | 9      | 9       | 9     | 9      |         |          |                | true
+---------+--------+---------+-------+--------+---------+----------+----------------+---------
```

5.3.6 `class`

Usage: `:class:<TableScheme-classname>:<scheme-spec>`

Uses an instance of a named class that implements the `uk.ac.starlink.table.TableScheme` interface and that has a no-arg constructor. Arguments to be passed to an instance of the named class are appended after a colon following the classname.

For example, the specification "`:class:uk.ac.starlink.table.LoopTableScheme:10`" would return a table constructed by the code `new uk.ac.starlink.table.LoopTableScheme().createTable("10")`.

Example:

```
:class:uk.ac.starlink.table.LoopTableScheme:5
+---+
| i |
+---+
| 0 |
| 1 |
| 2 |
| 3 |
| 4 |
+---+
```

6 Table Pipelines

Several of the tasks available in STILTS take one or more input tables, do something or other with them, and produce one or more output tables. This is a pretty obvious way to go about things, and in the most straightforward case that's exactly what happens: you name one or more input tables, specify the processing parameters, and name an output table; the task then reads the input tables from disk, does the processing and writes the output table to disk.

However, many of the tasks in STILTS allow you to do pre-processing of the input tables before the main job, post-processing of the output table after the main job, and to decide what happens to the final tabular result, without any intermediate storage of the data. Examples of the kind of pre-processing you might want to do are to rearrange the columns so that they have the right units for the main task, or replace 'magic' values such as -999 with genuine blank values; the kind of post-processing you might want to do is to sort the rows in the output table or delete some of the columns you're not interested in. As for the destination of the final table, you might want to write it to disk, but equally you might not want to store it anywhere, but only be interested in counting the number of rows, or seeing the minima/maxima of a few of the columns, or you might want to send it straight to TOPCAT or some other table viewing application for interactive analysis.

Clearly, you could achieve the same effect by running multiple applications: preprocess your original input tables to write intermediate files on disk, run the main processing application which reads those files from disk and writes a new output file, run another application to postprocess the output file and write a new final output file, and finally do something with this such as counting the rows in it or viewing it in TOPCAT. However, by doing it all within a single task instead, no intermediate results have to be stored, and the whole sequence can be very much more efficient. You can think of this (if it helps) like a Unix pipeline, except what is being streamed from the start to the end of the pipe is not bytes, but table metadata and data. In most cases, the table data is streamed through the pipeline a row at a time, meaning that the amount of memory required is small (though in some cases, for instance row sorting and crossmatching, this is not possible).

Tasks which allow this pre/post-processing, or "filtering", have parameters with names like "`cmd`" which you use to specify processing steps. Tasks with multiple input tables (`tmatch2`, `tskymatch2`, `tcatn`, `tjoin`) may have parameters named `icmd1`, `icmd2`, ... for preprocessing the different input tables and `ocmd` for postprocessing the output table. `tpipe` does nothing except filtering, so there is no distinction between pre- and post-processing, and its filter parameter is just named `cmd`. `tpipe` additionally has a `script` parameter which allows you to use a text file to write the commands in, to prevent the command line getting too long. In both cases there is a parameter named `omode` which defines the "output mode", that is, what happens to the post-processed output table that comes out of the end of the pipeline.

Section 6.1 lists the processing steps available, and explains how to use them, Section 6.2 and Section 6.3 describe the syntax used in some of these filter commands for specifying columns, and Section 6.4 describes the available output modes. See the examples in the command reference, and particularly the `tpipe` examples (Appendix B.39.2), for some examples putting all this together.

6.1 Processing Filters

This section lists the filter commands which can be used for table pipeline processing, in conjunction with `cmd`- or `script`-type parameters.

You can string as many of these together as you like. On the command line, you can repeat the `cmd` (or `icmd1`, or `ocmd`...) parameter multiple times, or use one `cmd` parameter and separate different filter specifiers with semicolons ("`;`"). The effect is the same.

It's important to note that each command in the sequence of processing steps acts on the table at that

point in the sequence. Thus either of the two identical invocations:

```
stilts tpipe cmd='delcols 1; delcols 1; delcols 1'
stilts tpipe cmd='delcols 1' cmd='delcols 1' cmd='delcols 1'
```

has the same effect as

```
stilts tpipe cmd='delcols "1 2 3"'
```

since in the first case the columns are shifted left after each one is deleted, so the table seen by each step has one fewer column than the one before. Note also the use of quotes in the latter of the examples above, which is necessary so that the `<colid-list>` of the `delcols` command is interpreted as one argument not three separate words.

The available filters are described in the following subsections.

6.1.1 `addcol`

Usage:

```
addcol [-after <col-id> | -before <col-id>]
       [-units <units>] [-ucd <ucd>] [-utype <utype>] [-desc <descrip>]
       [-shape <n>[,<n>...][,*]] [-elsize <n>]
       <col-name> <expr>
```

Add a new column called `<col-name>` defined by the algebraic expression `<expr>`. By default the new column appears after the last column of the table, but you can position it either before or after a specified column using the `-before` or `-after` flags respectively.

The `-units`, `-ucd`, `-utype` and `-desc` flags can be used to define textual metadata values for the new column.

The `-shape` flag can also be used, but is intended only for array-valued columns, e.g. `-shape 3,3` to declare a 3x3 array. The final entry only in the shape list may be a "`*`" character to indicate unknown extent. Array values with no specified shape effectively have a shape of "`*`". The `-elsize` flag may be used to specify the length of fixed length strings; use with non-string columns is not recommended.

Syntax for the `<expr>` and `<col-id>` arguments is described in the manual.

6.1.2 `addpixsample`

Usage:

```
addpixsample [-radius <expr-rad>] [-systems <in-sys> <pix-sys>]
             <expr-lon> <expr-lat> <healpix-file>
```

Samples pixel data from an all-sky image file in HEALPix format. The `<healpix-file>` argument must be the filename of a table containing HEALPix pixel data. The URL of such a file can be used instead, but local files are likely to be more efficient.

The `<expr-lon>` and `<expr-lat>` arguments give expressions for the longitude and latitude in degrees for each row of the input table; this is usually just the column names. The long/lat must usually be in the same coordinate system as that used for the HEALPix data, so if the one is in galactic coordinates the other must be as well. If this is not the case, use the `-systems` flag to give the input long/lat and healpix data coordinate system names respectively. The available coordinate system names are:

- `icrs`: ICRS (Right Ascension, Declination)

- `fk5`: FK5 J2000.0 (Right Ascension, Declination)
- `fk4`: FK4 B1950.0 (Right Ascension, Declination)
- `galactic`: IAU 1958 Galactic (Longitude, Latitude)
- `supergalactic`: de Vaucouleurs Supergalactic (Longitude, Latitude)
- `ecliptic`: Ecliptic (Longitude, Latitude)

The `<expr-rad>`, if present, is a constant or expression giving the radius in degrees over which pixels will be averaged to obtain the result values. Note that this averaging is somewhat approximate; pixels partly covered by the specified disc are weighted the same as those fully covered. If no radius is specified, the value of the pixel covering the central position will be used.

The `<healpix-file>` file is a table with one row per HEALPix pixel and one or more columns representing pixel data. A new column will be added to the output table corresponding to each of these pixel columns. This type of data is available in FITS tables for a number of all-sky data sets, particularly from the LAMBDA (http://lambda.gsfc.nasa.gov/) archive; see for instance the page on foreground products (including dust emission, reddening etc) or WMAP 7 year data. If the filename given does not appear to point to a file of the appropriate format, an error will result. Note the LAMBDA files mostly (all?) use galactic coordinates, so coordinate conversion using the `-systems` flag may be appropriate, see above.

Syntax for the `<expr-lon>` , `<expr-lat>` and `<expr-rad>` arguments is described in the manual.

This filter is somewhat experimental, and its usage may be changed or replaced in a future version.

Note: you may prefer to use the `pixsample` command instead.

6.1.3 `addresolve`

Usage:

```
addresolve <col-id-objname> <col-name-ra> <col-name-dec>
```

Performs name resolution on the string-valued column `<col-id-objname>` and appends two new columns `<col-name-ra>` and `<col-name-dec>` containing the resolved Right Ascension and Declination in degrees.

Syntax for the `<col-id-objname>` argument is described in Section 6.2.

UCDs are added to the new columns in a way which tries to be consistent with any UCDs already existing in the table.

Since this filter works by interrogating a remote service, it will obviously be slow. The current implementation is experimental; it may be replaced in a future release by some way of doing the same thing (perhaps a new STILTS task) which is able to work more efficiently by dispatching multiple concurrent requests.

This is currently implemented using the Simbad service operated by CDS.

6.1.4 `addskycoords`

Usage:

```
addskycoords [-epoch <expr>] [-inunit deg|rad|sex] [-outunit deg|rad|sex]
             <insys> <outsys> <col-id1> <col-id2> <col-name1> <col-name2>
```

Add new columns to the table representing position on the sky. The values are determined by

converting a sky position whose coordinates are contained in existing columns. The `<col-id>` arguments give identifiers for the two input coordinate columns in the coordinate system named by `<insys>`, and the `<col-name>` arguments name the two new columns, which will be in the coordinate system named by `<outsys>`. The `<insys>` and `<outsys>` coordinate system specifiers are one of

- `icrs`: ICRS (Right Ascension, Declination)
- `fk5`: FK5 J2000.0 (Right Ascension, Declination)
- `fk4`: FK4 B1950.0 (Right Ascension, Declination)
- `galactic`: IAU 1958 Galactic (Longitude, Latitude)
- `supergalactic`: de Vaucouleurs Supergalactic (Longitude, Latitude)
- `ecliptic`: Ecliptic (Longitude, Latitude)

The `-inunit` and `-outunit` flags may be used to indicate the units of the existing coordinates and the units for the new coordinates respectively; use one of `degrees`, `radians` or `sexagesimal` (may be abbreviated), otherwise degrees will be assumed. For sexagesimal, the two corresponding columns must be string-valued in forms like hh:mm:ss.s and dd:mm:ss.s respectively.

For certain conversions, the value specified by the `-epoch` flag is of significance. Where significant its value defaults to 2000.0.

Syntax for the `<expr>` , `<col-id1>` and `<col-id2>` arguments is described in the manual.

6.1.5 `assert`

Usage:

```
assert <test-expr> [<msg-expr>]
```

Check that a boolean expression is true for each row. If the expression `<test-expr>` does not evaluate true for any row of the table, execution terminates with an error. As long as no error occurs, the output table is identical to the input one.

If the `<msg-expr>` parameter is supplied, then on failure it will be evaluated and its value presented in the error message.

The exception generated by an assertion violation is of class `uk.ac.starlink.ttools.filter.AssertException` although that is not usually obvious if you are running from the shell in the usual way.

Syntax for the `<test-expr>` and `<msg-expr>` arguments is described in the manual.

6.1.6 `badval`

Usage:

```
badval <bad-val> <colid-list>
```

For each column specified in `<colid-list>` any occurrence of the value `<bad-val>` is replaced by a blank entry.

Syntax for the `<colid-list>` argument is described in Section 6.3.

6.1.7 `cache`

Usage:

```
cache
```

Stores in memory or on disk a temporary copy of the table at this point in the pipeline. This can provide improvements in efficiency if there is an expensive step upstream and a step which requires more than one read of the data downstream. If you see an error like "Can't re-read data from stream" then adding this step near the start of the filters might help.

The output table contains no code-level reference to the input table, so this filter can also be useful when managing tables that have become deeply nested as the result of successively applying many STILTS operations.

The result of this filter is guaranteed to be random-access.

See also the `random` filter, which caches only when the input table is not random-access.

6.1.8 `check`

Usage:

```
check
```

Runs checks on the table at the indicated point in the processing pipeline. This is strictly a debugging measure, and may be time-consuming for large tables.

6.1.9 `clearparams`

Usage:

```
clearparams <pname> ...
```

Clears the value of one or more named parameters. Each of the `<pname>` values supplied may be either a parameter name or a simple wildcard expression matching parameter names. Currently the only wildcarding is a "`*`" to match any sequence of characters. `clearparams *` will clear all the parameters in the table.

It is not an error to supply `<pname>`s which do not exist in the table - these have no effect.

6.1.10 `collapsecols`

Usage:

```
collapsecols [-[no]keepscalars] <array-colname> <col-id0> <ncol>
```

Adds a new array-valued column by using the values from a specified range of scalar columns as array elements. The new column is named `<array-colname>`, and produced from the sequence of `<ncol>` scalar columns starting with `<col-id0>`.

The array type of the output column is determined by the type of the first input column (`<col-id0>`). If it is of type `Double`, the output array column will be a `double[]` array, and similarly for types `Long`, `Integer`, `Float` and `Boolean`. Other integer types are currently mapped to `int[]`, and object types, e.g. `String`, to the corresponding array type. Array elements for null or mistyped input values are mapped to NaN for floating point types, but ***note*** that they currently just turn into zeros for integer array types and `false` for boolean.

By default the scalar columns that have been used are removed from the output table and the new column replaces them at the same position. However, if you supply the `-keepscalars` flag they will

be retained alongside the new array column (the new column will appear just after the run of scalar columns).

This filter does the opposite of `explodecols`.

Syntax for the `<col-id0>` argument is described in Section 6.2.

6.1.11 `colmeta`

Usage:

```
colmeta [-name <name>] [-units <units>] [-ucd <ucd>] [-utype <utype>]
        [-desc <descrip>] [-shape <n>[,<n>...][,*]] [-elsize <n>]
        <colid-list>
```

Modifies the metadata of one or more columns. Some or all of the name, units, ucd, utype, description, shape and elementsize of the column(s), identified by `<colid-list>` can be set by using some or all of the listed flags. Typically, `<colid-list>` will simply be the name of a single column.

The `-name`, `-units`, `-ucd`, `-utype` and `-desc` flags just take textual arguments. The `-shape` flag can also be used, but is intended only for array-valued columns, e.g. `-shape 3,3` to declare a 3x3 array. The final entry only in the shape list may be a "`*`" character to indicate unknown extent. Array values with no specified shape effectively have a shape of "`*`". The `-elsize` flag may be used to specify the length of fixed length strings; use with non-string columns is not recommended.

Syntax for the `<colid-list>` argument is described in Section 6.3.

6.1.12 `delcols`

Usage:

```
delcols <colid-list>
```

Delete the specified columns. The same column may harmlessly be specified more than once.

Syntax for the `<colid-list>` argument is described in Section 6.3.

6.1.13 `every`

Usage:

```
every [-exact|-approx] <step>
```

Include only every `<step>`'th row in the result, starting with the first row. The optional `-approx`/`-exact` argument controls whether the selection needs to be exact; in some cases an approximate calculation can take advantage of parallelism where an exact one cannot.

6.1.14 `explodeall`

Usage:

```
explodeall [-ifndim <ndim>] [-ifshape <dims>]
```

Replaces any columns which is an N-element arrays with N scalar columns. Only columns with fixed array sizes are affected. The action can be restricted to only columns of a certain shape using the flags.

If the `-ifndim` flag is used, then only columns of dimensionality `<ndim>` will be exploded. `<ndim>` may be 1, 2, ...

If the `-ifshape` flag is used, then only columns with a specific shape will be exploded; `<dims>` is a space- or comma-separated list of dimension extents, with the most rapidly-varying first, e.g. `'2 5'` to explode all 2 x 5 element array columns.

6.1.15 `explodecols`

Usage:

```
explodecols <colid-list>
```

Takes a list of specified columns which represent N-element arrays and replaces each one with N scalar columns. Each of the columns specified by `<colid-list>` must have a fixed-length array type, though not all the arrays need to have the same number of elements.

This filter does the opposite of `collapsecols`.

Syntax for the `<colid-list>` argument is described in Section 6.3.

6.1.16 `fixcolnames`

Usage:

```
fixcolnames
```

Renames all columns and parameters in the input table so that they have names which have convenient syntax for STILTS. For the most part this means replacing spaces and other non-alphanumeric characters with underscores. This is a convenience which lets you use column names in algebraic expressions and other STILTS syntax.

6.1.17 `head`

Usage:

```
head <nrows>
```

Include only the first `<nrows>` rows of the table. If the table has fewer than `<nrows>` rows then it will be unchanged.

6.1.18 `healpixmeta`

Usage:

```
healpixmeta [-level <n>] [-implicit|-column <col-id>] [-csys C|G|E] [-nested|-ring]
```

Adjusts the table metadata items that describe how HEALPix pixel data is encoded in the table.

Zero or more of the following flags may be supplied:

- `-level <n>`: Defines the HEALPix level; the sky is split into 12*4^n pixels. This quantity is equal to logarithm base 2 of NSIDE.
- `-implicit`: Declares that pixel indices are implicit, so that row *i* represents HEALPix pixel index *i*. The table should have 12*4^level rows in this case. Not to be used with `-column`.
- `-column <col-id>`: Declares that the column identified contains the (0-based) HEALPix pixel

index. Not to be used with `-implicit`.
- `-csys C|G|E`: Declares the sky coordinate system to which the HEALPix pixels apply: **C**elestial(=equatorial), **G**alactic or **E**cliptic. Some applications assume Galactic if this is not specified.
- `-nested`: Declares that the NESTED ordering scheme is in use. Not to be used with `-ring`.
- `-ring`: Declares that the RING ordering scheme is in use. Not to be used with `-nested`.

The effect of this filter is to write, or overwrite, certain special table parameters (per-table metadata) that STIL uses to describe how HEALPix pixel information is encoded in a table, specifically the HEALPix level, the column containing pixel index, the ordering scheme, and the sky coordinate system. Adding these parameters doesn't do anything on its own, but some of the STIL I/O handlers recognise these parameters, and they affect how the table will be formatted for output. In particular, if you set these parameters and then output to FITS format, the output table will contain headers defined by the HEALPix-FITS serialization format which is understood by several other applications to describe HEALPix maps. If you write to VOTable format, the metadata will only be recognised by other STIL-based applications but it means that if you, e.g., load the table into TOPCAT and then write it out again as FITS, the HEALPix information should be preserved.

When writing tables marked up like this to FITS, you have two options. If you write to one of the "normal" FITS formats (e.g. `fits`, `fits-basic`) then suitable headers will be added; in this case if an explicit pixel index column is used it must be the first column, and should be named "PIXEL". This may be enough for other applications to recognise the HEALPix metadata. However, if you use the special `fits-healpix` format more efforts will be made to conform to the HEALPix-FITS convention, for instance moving and renaming the explicit pixel index column if required.

The table parameters affected by this filter are: `STIL_HPX_LEVEL`, `STIL_HPX_ISNEST`, `STIL_HPX_COLNAME`, `STIL_HPX_CSYS`. Note these are not defined by any standard, they are defined and used only by STILTS and related applications (TOPCAT).

Syntax for the `<col-id>` argument is described in Section 6.2.

6.1.19 `keepcols`

Usage:

```
keepcols <colid-list>
```

Select the columns from the input table which will be included in the output table. The output table will include only those columns listed in `<colid-list>`, in that order. The same column may be listed more than once, in which case it will appear in the output table more than once.

Syntax for the `<colid-list>` argument is described in Section 6.3.

6.1.20 `meta`

Usage:

```
meta [<item> ...]
```

Provides information about the metadata for each column. This filter turns the table sideways, so that each row of the output corresponds to a column of the input. The columns of the output table contain metadata items such as column name, units, UCD etc corresponding to each column of the input table.

By default the output table contains columns for all metadata items for which any of the columns have non-blank values.

However, the output may be customised by supplying one or more `<item>` headings, in which case exactly those columns will appear, regardless of whether they have entries. It is not an error to specify an item for which no metadata exists in any of the columns (such entries will result in empty columns).

Some of the metadata items commonly found are:

- `Index`: Position of column in table
- `Name`: Column name
- `Class`: Data type of objects in column
- `Shape`: Shape of array values
- `ElSize`: Size of each element in column (mostly useful for strings)
- `Units`: Unit string
- `Description`: Description of data in the column
- `UCD`: Unified Content Descriptor
- `Utype`: Type in data model
- `Xtype`: Extended data type
- `CoosysSystem`: Sky coordinate system name from COOSYS
- `CoosysEpoch`: Sky epoch from COOSYS
- `CoosysEquinox`: Sky equinox from COOSYS
- `TimesysTimeorigin`: Time origin from TIMESYS
- `TimesysTimescale`: Timescale from TIMESYS
- `TimesysRefposition`: Ref position from TIMESYS
- `STIL_HPX_LEVEL`: Level of HEALPix pixels contained in the table (nside=2^level)
- `STIL_HPX_ISNEST`: True for NEST indexation scheme, False for RING
- `STIL_HPX_COLNAME`: Name of the table column containing HEALPix index; null value or empty string indicates implicit
- `STIL_HPX_CSYS`: 'C'=celestial/equatorial, 'G'=galactic, 'E'=ecliptic

Any table parameters of the input table are propagated to the output one.

6.1.21 `progress`

Usage:

```
progress
```

Monitors progress by displaying the number of rows processed so far on the terminal (standard error). This number is updated every second or thereabouts; if all the processing is done in under a second you may not see any output. If the total number of rows in the table is known, an ASCII-art progress bar is updated, otherwise just the number of rows seen so far is written.

Note under some circumstances progress may appear to complete before the actual work of the task is done since part of the processing involves slurping up the whole table to provide random access on it. In this case, applying the `cache` upstream may help.

6.1.22 `random`

Usage:

```
random
```

Ensures that random access is available on this table. If the table currently has random access, it has no effect. If only sequential access is available, the table is cached so that downstream steps will see the cached, hence random-access, copy.

6.1.23 `randomview`

Usage:

```
randomview
```

Ensures that steps downstream only use random access methods for table access. If the table is sequential only, this will result in an error. Only useful for debugging.

6.1.24 `repeat`

Usage:

```
repeat [-row|-table] <count>
```

Repeats the rows of a table multiple times to produce a longer table. The output table will have `<count>` times as many rows as the input table.

The optional flag determines the sequence of the output rows. If `<count>`=2 and there are three rows, the output sequence will be 112233 for `-row` and 123123 for `-table`. The default behaviour is currently `-table`.

The `<count>` value will usually be a constant integer value, but it can be an expression evaluated in the context of the table.

6.1.25 `replacecol`

Usage:

```
replacecol [-name <name>] [-units <units>] [-ucd <ucd>] [-utype <utype>]
           [-desc <descrip>]
           <col-id> <expr>
```

Replaces the content of a column with the value of an algebraic expression. The old values are discarded in favour of the result of evaluating `<expr>`. You can specify the metadata for the new column using the `-name`, `-units`, `-ucd`, `-utype` and `-desc` flags; for any of these items which you do not specify, they will take the values from the column being replaced.

It is legal to reference the replaced column in the expression, so for example "`replacecol pixsize pixsize*2`" just multiplies the values in column `pixsize` by 2.

Syntax for the `<col-id>` and `<expr>` arguments is described in the manual.

6.1.26 `replaceval`

Usage:

```
replaceval <old-val> <new-val> <colid-list>
```

For each column specified in `<colid-list>` any instance of `<old-val>` is replaced by `<new-val>`. The value string '`null`' can be used for either `<old-value>` or `<new-value>` to indicate a blank value (but see also the `badval` filter).

Syntax for the `<colid-list>` argument is described in Section 6.3.

6.1.27 `rowrange`

Usage:

```
rowrange <first> <last>|+<count>
```

Includes only rows in a given range. The range can either be supplied as "`<first> <last>`", where row indices are inclusive, or "`<first> +<count>`". In either case, the first row is numbered 1.

Thus, to get the first hundred rows, use either "`rowrange 1 100`" or "`rowrange 1 +100`" and to get the second hundred, either "`rowrange 101 200`" or "`rowrange 101 +100`"

6.1.28 `select`

Usage:

```
select <expr>
```

Include in the output table only rows for which the expression `<expr>` evaluates to true. `<expr>` must be an expression which evaluates to a boolean value (true/false).

Syntax for the `<expr>` argument is described in Section 10.

6.1.29 `seqview`

Usage:

```
seqview
```

Ensures that steps downstream see the table as sequential access. Any attempts at random access will fail. Only useful for debugging.

6.1.30 `setparam`

Usage:

```
setparam [-type byte|short|int|long|float|double|boolean|string]
         [-desc <descrip>] [-unit <units>] [-ucd <ucd>] [-utype <utype>]
         <pname> <pexpr>
```

Sets a named parameter in the table to a given value. The parameter named `<pname>` is set to the value `<pexpr>`, which may be a literal value or an expression involving mathematical operations and other parameter names (using the `param$<name>` syntax). By default, the data type of the parameter is determined by the type of the supplied expression, but this can be overridden using the `-type` flag. The parameter description, units, UCD and Utype attributes may optionally be set using the other flags.

6.1.31 `sort`

Usage:

```
sort [-down] [-nullsfirst] [-[no]parallel] <key-list>
```

Sorts the table according to the value of one or more algebraic expressions. The sort key expressions appear, as separate (space-separated) words, in `<key-list>`; sorting is done on the first expression first, but if that results in a tie then the second one is used, and so on.

Each expression must evaluate to a type that it makes sense to sort, for instance numeric. If the `-down` flag is used, the sort order is descending rather than ascending.

Blank entries are by default considered to come at the end of the collation sequence, but if the `-nullsfirst` flag is given then they are considered to come at the start instead.

By default, sorting is done sequentially for small tables and in parallel for large tables, but this can be controlled with the `-parallel` or `-noparallel` flag.

Syntax for the `<key-list>` argument is described in Section 10.

6.1.32 `sorthead`

Usage:

```
sorthead [-tail] [-down] [-nullsfirst] <nrows> <key-list>
```

Performs a sort on the table according to the value of one or more algebraic expressions, retaining only `<nrows>` rows at the head of the resulting sorted table. The sort key expressions appear, as separate (space-separated) words, in `<key-list>`; sorting is done on the first expression first, but if that results in a tie then the second one is used, and so on. Each expression must evaluate to a type that it makes sense to sort, for instance numeric.

If the `-tail` flag is used, then the last `<nrows>` rows rather than the first ones are retained.

If the `-down` flag is used the sort order is descending rather than ascending.

Blank entries are by default considered to come at the end of the collation sequence, but if the `-nullsfirst` flag is given then they are considered to come at the start instead.

This filter is functionally equivalent to using `sort` followed by `head`, but it can be done in one pass and is usually cheaper on memory and faster, as long as `<nrows>` is significantly lower than the size of the table.

Syntax for the `<key-list>` argument is described in Section 10.

6.1.33 `stats`

Usage:

```
stats [-[no]parallel] [-qapprox|-qexact] [<item> ...]
```

Calculates statistics on the data in the table. This filter turns the table sideways, so that each row of the output corresponds to a column of the input. The columns of the output table contain statistical items such as mean, standard deviation etc corresponding to each column of the input table.

By default the output table contains columns for the following items:

- `Name`: Column name
- `Mean`: Average
- `StDev`: Population Standard deviation
- `Minimum`: Numeric minimum
- `Maximum`: Numeric maximum
- `NGood`: Number of non-blank cells

However, the output may be customised by supplying one or more `<item>` headings. These may be selected from the above as well as the following:

- `NBad`: Number of blank cells
- `Variance`: Population Variance

- `SampStDev`: Sample Standard Deviation
- `SampVariance`: Sample Variance
- `MedAbsDev`: Median Absolute Deviation
- `ScMedAbsDev`: Median Absolute Deviation * 1.4826
- `Skew`: Gamma 1 skewness measure
- `Kurtosis`: Gamma 2 peakedness measure
- `Sum`: Sum of values
- `MinPos`: Row index of numeric minimum
- `MaxPos`: Row index of numeric maximum
- `Cardinality`: Number of distinct values in column; values >100 ignored
- `Median`: Middle value in sequence
- `Quartile1`: First quartile
- `Quartile2`: Second quartile
- `Quartile3`: Third quartile
- `ArrayNGood`: Per-element non-blank counts for fixed-length array columns
- `ArraySum`: Per-element sums for fixed-length array columns
- `ArrayMean`: Per-element means for fixed-length array columns
- `ArrayStDev`: Per-element population standard deviation for fixed-length array columns

Additionally, the form "Q.*nn*" may be used to represent the quantile corresponding to the proportion 0.*nn*, e.g.:

- `Q.25`: First quartile
- `Q.625`: Fifth octile

Any parameters of the input table are propagated to the output one.

The `-qapprox` or `-qexact` flag controls how quantiles are calculated. With `-qexact` they are calculated exactly, but this requires memory usage scaling with the number of rows. If the `-qapprox` flag is supplied, an method is used which is typically slower and produces only approximate values, but which will work in fixed memory and so can be used for arbitrarily large tables. By default, exact calculation is used. These flags are ignored if neither quantiles nor the MAD are being calculated

The `-noparallel` flag may be supplied to inhibit multi-threaded statistics accumulation. Calculation is done in parallel by default if multi-threaded hardware is available, and it's usually faster.

6.1.34 `tablename`

Usage:

```
tablename <name>
```

Sets the table's name attribute to the given string.

6.1.35 `tail`

Usage:

```
tail <nrows>
```

Include only the last `<nrows>` rows of the table. If the table has fewer than `<nrows>` rows then it will be unchanged.

6.1.36 `transpose`

Usage:

```
transpose [-namecol <col-id>]
```

Transposes the input table so that columns become rows and vice versa. The `-namecol` flag can be used to specify a column in the input table which will provide the column names for the output table. The first column of the output table will contain the column names of the input table.

Syntax for the `<col-id>` argument is described in Section 6.2.

6.1.37 `uniq`

Usage:

```
uniq [-count] [<colid-list>]
```

Eliminates adjacent rows which have the same values. If used with no arguments, then any row which has identical values to its predecessor is removed.

If the `<colid-list>` parameter is given then only the values in the specified columns must be equal in order for the row to be removed.

If the `-count` flag is given, then an additional column with the name DupCount will be prepended to the table giving a count of the number of duplicated input rows represented by each output row. A unique row has a DupCount value of 1.

Syntax for the `<colid-list>` argument is described in Section 6.3.

6.2 Specifying a Single Column

If an argument is specified in the help text for a command with the symbol `<col-id>` it means you must give a string which identifies one of the existing columns in a table.

There are several ways you can specify a column in this context:

Column Name
The name of the column may be used if it contains no spaces. It is usually matched case insensitively. If multiple columns have the same name, the first one that matches is selected.

Column Index or $ID
The index of the column may always be used; this is a useful fallback if the column name isn't suitable for some reason. The first column is '1', the second is '2' and so on. You may alternatively use the forms '$1', '$2' etc.

Tip: if counting which column has which index is giving you a headache, running `tpipe` with `omode=meta` or `omode=stats` on the table may help.

Column ucd$ specifier
If the column has a Unified Content Descriptor (this will usually only be the case for VOTable or possibly FITS format tables) you can refer to it using an identifier of the form "`ucd$<ucd-spec>`". Depending on the version of UCD scheme used, UCDs can contain various punctuation marks such as underscores, semicolons and dots; for the purpose of this syntax these should all be represented as underscores ("_"). So to identify a column which has the UCD "`phot.mag;em.opt.R`", you should use the identifier "`ucd$phot_mag_em_opt_r`". Matching is not case-sensitive. Futhermore, a trailing underscore acts as a wildcard, so that the above column could also be referenced using the identifier "`ucd$phot_mag_`". If multiple columns have UCDs which match the given identifer, the first one will be used.

Column utype$ specifier

If the column has a **Utype** (this will usually only be the case for VOTable or possibly FITS format tables) you can refer to it using an identifier of the form "`utype$<utype-spec>`". Utypes may contain various punctuation marks such as colons and dots; for the purpose of this syntax these should all be represented as underscores ("_"). So to identify a column which has the Utype "`ssa:Access.Format`", you should use the identifier "`utype$ssa_Access_format`". Matching is not case-sensitive. If multiple columns have Utypes which match the given identifier, the first one will be used.

6.3 Specifying a List of Columns

If an argument is specified in the help text for a command with the symbol `<colid-list>` it means you must give a string which identifies a list of zero, one or more of the existing columns in a table. The string you specify is a separated into separate tokens by whitespace, which means that you will normally have to surround it in single or double quotes to ensure that it is treated as a single argument and not several of them.

Each token in the `<colid-list>` string may be one of the following:

Single Column Identifier

The identifier for a single column, as described in Section 6.2: one of `<name>`, `<index>`, `$<index>`, `ucd$<ucd-expr>` or `utype$<utype-expr>`.

Wildcard Expression

You can use a simple form of wildcard expression which expands to any columns in the table whose names match the pattern. Currently, the only special character is an asterisk '`*`' which matches any sequence of characters. To match an unknown sequence at the start or end of the string an asterisk must be given explicitly. Other than that, matching is usually case insensitive. The order of the expanded list is the same as the order in which the columns appear in the table.

Thus "`col*`" will match columns named `col1`, `Column2` and `COL_1024`, but not `decOld`. "`*MAG*`" will match columns named `magnitude`, `ABS_MAG_U` and `JMAG`. "`*`" on its own expands to a list of all the columns of the table in order.

Column Range

You can specify a range of columns in order using a token of the form `<first-colid>-<last-colid>`, where the syntax for `<first-colid>` and `<last-colid>` is a **Single Column Identifier** as above (as long as it doesn't contain a "`-`" character). The range is inclusive, so the first and last column are both included. The `<first-colid>` or `<last-colid>` part (but not both) may be omitted, to indicate all the columns from the start or all the columns to the end, respectively.

So "`RA-PARALLAX`" means all the columns starting with the one named `RA` and ending with the one named `PARALLAX` (inclusive); "`1-100`" or "`$1-$100`" means the first hundred columns in the table; "`101-`" or "`$101-`" means all the columns apart from the first hundred, "`PARALLAX-`" means `PARALLAX` and all subsequent columns, etc.

Specifying a list which contains a given column more than once is not usually an error, but what effect it has depends on the function you are executing.

6.4 Output Modes

This section lists the output modes which can be used as the value of the `omode` parameter of `tpipe` and other commands. Typically, having produced a result table by pipeline processing an input one, you will write it out by specifying `omode=out` (or not using the `omode` parameter at all - `out` is the

default). However, you can do other things such as calculate statistics, display metadata, etc. In some of these cases, additional parameters are required. The different output modes, with their associated parameters, are described in the following subsections.

6.4.1 `cgi`

Usage:

```
omode=cgi ofmt=<out-format>
```

Writes a table to standard output in a way suitable for use as output from a CGI (Common Gateway Interface) program. This is very much like `out` mode but a short CGI header giving the MIME Content-Type is prepended to the output

Additional parameters for this output mode are:

`ofmt = <out-format>` ***(String)***
Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters).

[Default: `votable`]

6.4.2 `checksum`

Usage:

```
omode=checksum
```

Calculates a checksum from all the data in the table. The checksum is written to standard output in hexadecimal; row and column counts are also written.

If two tables have the same checksum it is extremely likely that they contain the same cell data. If they have a different checksum, their cell data differs. By default, the checksum implementation uses Adler32, which is fast but not cryptographically secure.

6.4.3 `count`

Usage:

```
omode=count
```

Counts the number of rows and columns and writes the result to standard output.

6.4.4 `discard`

Usage:

```
omode=discard
```

Reads all the data in the table in sequential mode and discards it. May be useful in conjunction with the `assert` filter.

6.4.5 `gui`

Usage:

```
omode=gui
```

Displays the table in a scrollable window.

6.4.6 `meta`

Usage:

```
omode=meta
```

Prints the table metadata to standard output. The name and type etc of each column is tabulated, and table parameters are also shown.

See the `meta` filter for more flexible output of table metadata.

6.4.7 `out`

Usage:

```
omode=out out=<out-table> ofmt=<out-format>
```

Writes a new table.

Additional parameters for this output mode are:

`out = <out-table>` ***(TableConsumer)***
The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

[Default: -]

`ofmt = <out-format>` ***(String)***
Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

[Default: `(auto)`]

6.4.8 `plastic`

Usage:

```
omode=plastic transport=string|file client=<app-name>
```

Broadcasts the table to any registered Plastic-aware applications. PLASTIC, the PLatform for AStronomical Tool InterConnection, is a tool interoperability protocol. A *Plastic hub* must be running in order for this to work.

Additional parameters for this output mode are:

`transport = string|file` ***(String)***
Determines the method (PLASTIC *message*) used to perform the PLASTIC communication. The choices are

- `string`: VOTable serialized as a string and passed as a call parameter (`ivo://votech.org/votable/load`). Not suitable for very large files.
- `file`: VOTable written to a temporary file and the filename passed as a call parameter

ivo://votech.org/votable/loadFromURL). The file ought to be deleted once it has been loaded. Not suitable for inter-machine communication.

If no value is set (null) then a decision will be taken based on the apparent size of the table.

client = <app-name> ***(String)***
Gives the name of a PLASTIC listener application which is to receive the broadcast table. If a non-null value is given, then only the first registered application which reports its application name as that value will receive the message. If no value is supplied, the broadcast will be to all listening applications.

6.4.9 samp

Usage:

```
omode=samp format=<value> client=<name-or-id>
```

Sends the table to registered SAMP-aware applications subscribed to a suitable table load MType. SAMP, the Simple Application Messaging Protocol, is a tool interoperability protocol. A *SAMP Hub* must be running for this to work.

Additional parameters for this output mode are:

format = <value> ***(String[])***
Gives one or more table format types for attempting the table transmission over SAMP. If multiple values are supplied, they should be separated by spaces. Each value supplied for this parameter corresponds to a different MType which may be used for the transmission. If a single value is used, a SAMP broadcast will be used. If multiple values are used, each registered client will be interrogated to see whether it subscribes to the corresponding MTypes in order; the first one to which it is subscribed will be used to send the table. The standard options are

- votable: use MType table.load.votable
- fits: use MType table.load.fits

If any other string is used which corresponds to one of STILTS's known table output formats, an attempt will be made to use an ad-hoc MType of the form table.load.format.

[Default: votable fits]

client = <name-or-id> ***(String)***
Identifies a registered SAMP client which is to receive the table. Either the client ID or the (case-insensitive) application name may be used. If a non-null value is given, then the table will be sent to only the first client with the given name or ID. If no value is supplied the table will be sent to all suitably subscribed clients.

6.4.10 stats

Usage:

```
omode=stats
```

Calculates and displays univariate statistics for each of the numeric columns in the table. The following entries are shown for each column as appropriate:

- mean
- population standard deviation
- minimum

- maximum
- number of non-null entries

See the `stats` filter for more flexible statistical calculations.

6.4.11 `topcat`

Usage:

```
omode=topcat
```

Attempts to display the output table directly in TOPCAT. If a TOPCAT instance is already running on the local host, an attempt will be made to open the table in that. A variety of mechanisms are used to attempt communication with an existing TOPCAT instance. In order:

1. SAMP using existing hub (TOPCAT v3.4+ only, requires SAMP hub to be running)
2. PLASTIC using existing hub (requires PLASTIC hub to be running)
3. SOAP (requires TOPCAT to run with somewhat deprecated `-soap` flag, may be limitations on table size)
4. SAMP using internal, short-lived hub (TOPCAT v3.4+ only, running hub not required, but may be slow. It's better to start an external hub, e.g. `topcat -exthub`)

Failing that, an attempt will be made to launch a new TOPCAT instance for display. This only works if the TOPCAT classes are on the class path.

If large tables are involved, starting TOPCAT with the `-disk` flag is probably a good idea.

6.4.12 `tosql`

Usage:

```
omode=tosql protocol=<jdbc-protocol> host=<value> db=<db-name>
            dbtable=<table-name> write=create|dropcreate|append
            user=<username> password=<passwd>
```

Writes a new table to an SQL database. You need the appropriate JDBC drivers and `-Djdbc.drivers` set as usual (see Section 3.4).

Additional parameters for this output mode are:

`protocol = <jdbc-protocol>` ***(String)***
The driver-specific sub-protocol specifier for the JDBC connection. For MySQL's Connector/J driver, this is `mysql`, and for PostgreSQL's driver it is `postgresql`. For other drivers, you may have to consult the driver documentation.

`host = <value>` ***(String)***
The host which is acting as a database server.

[Default: `localhost`]

`db = <db-name>` ***(String)***
The name of the database on the server into which the new table will be written.

The value of this parameter forms the last part of the JDBC connection URL. That means that for some JDBC drivers, you can append parameter specifications to it to affect the way the connection is made to the database, e.g. "`db=some_db?useSSL=false`" for MySQL's Connector/J.

`dbtable = <table-name>` ***(String)***
The name of the table which will be written to the database.

`write = create|dropcreate|append` ***(WriteMode)***

Controls how the values are written to a table in the database. The options are:

- `create`: Creates a new table before writing. It is an error if a table of the same name already exists.
- `dropcreate`: Creates a new database table before writing. If a table of the same name already exists, it is dropped first.
- `append`: Appends to an existing table. An error results if the named table has the wrong structure (number or types of columns) for the data being written.

[Default: `create`]

`user = <username>` ***(String)***

User name for the SQL connection to the database.

[Default: `mbt`]

`password = <passwd>` ***(String)***

Password for the SQL connection to the database.

7 Crossmatching

STILTS offers flexible and efficient facilities for crossmatching tables. Crossmatching is identifying different rows, which may be in the same or different tables, that refer to the same item. In an astronomical context such an item is usually, though not necessarily, an astronomical source or object. This operation corresponds to what in database terminology is called a *join*.

There are various complexities to specifying such a match. In the first place you have to define what is the condition that must be satisfied for two rows to be considered matching. In the second place you must decide what happens if, for a given row, more than one match can be found. Finally, you have to decide what to do having worked out what the matched rows are; the result will generally be presented as a new output table, but there are various choices about what columns and rows it will consist of. Some of these issues are discussed in this section, and others in the reference sections on the tools themselves in Appendix B.

Matching can in general be a computationally intensive process. The algorithm used by the `tmatch*` tasks in STILTS, except in pathological cases, scales as *O(N log(N))* or thereabouts, where *N* is the total number of rows in all the tables being matched. No preparation (such as sorting) is required on the tables prior to invoking the matching operation. It is reasonably fast; for instance an RA, Dec positional match of two 10^5-row catalogues takes of the order of 60 seconds on current (2005 laptop) hardware. Attempting matches with large tables can lead to running out of memory; the calculation just mentioned required a java heap size of around 200Mb (`-Xmx200M`).

In the current release of STILTS the following tasks are provided for crossmatching between local tables:

`tmatch2`
: Generic crossmatching between two tables.

`tskymatch2`
: Crossmatching between two tables where the matching criterion is a fixed separation on the sky. This is simply a stripped-down version of `tmatch2` provided for convenience when the full generality is not required.

`tmatch1`
: Generic crossmatching internal to a single table. The basic task this performs is to identify groups of rows within a single table which match each other.

`tmatchn`
: Generic crossmatching between multiple (>2) tables.

`tjoin`
: Trivial join operation between multiple tables in which no row re-ordering is required. This barely warrants the term "crossmatch" and the concepts explained in the rest of this section are not relevant to it.

7.1 Match Criteria

Determining whether one row represents the same item as another is done by comparing the values in certain of their columns to see if they are the same or similar. The most common astronomical case is to say that two rows match if their celestial coordinates (right ascension and declination) are within a given small radius of each other on the sky. There are other possibilities; for instance the coordinates to compare may be in a Cartesian space, or have a higher (or lower) dimensionality than two, or the match may be exact rather than within an error radius....

If you just need to match two tables according to sky position with fixed errors you are

recommended to use the simplified `tskymatch2` task. For other cases, this section describes how to specify much more flexible match criteria for use with `tmatch1`, `tmatch2` or `tmatchn` by setting the following parameters:

`matcher`
Name of the match criteria type.

`params`
Fixed value(s) giving the parameters of the match (typically an error radius). If more than one value is required, the values should be separated by spaces.

`values*`
Expressions to be compared between rows. This will typically contain the names of one or more columns, but each element may be an algebraic expression (see Section 10) rather than just a column name if required. If more than one value is required, the values should be separated by spaces. There is one of these parameters for each table taking part in the match, so for `tmatch2` you must specify both `values1` and `values2`.

`tuning`
Fixed value(s) supplying tuning parameters for the match algorithm. If there is more than one value, they should be separated by spaces. This value will have a sensible default, so you do not need to supply it, but providing adjusted values may make your match run faster or require less memory (or the reverse). Adjusting tuning parameters will not change the result of any match, only the resources required to run it. Looking at the progress output of a match will indicate what tuning values have been used; adjusting the value a bit up or down is a good way to experiment.

For example, suppose we wish to locate objects in two tables which are within 3 arcseconds of each other on the sky. One table has columns RA and DEC which give coordinates in degrees, and the other has columns RArad and DECrad which give coordinates in radians. These are the arguments which would be used to tell `tmatch2` what the match criteria are:

```
matcher=sky
params=3
values1='RA DEC'
values2='radiansToDegrees(RArad) radiansToDegrees(DECrad)'
```

It is clearly important that corresponding values are comparable (in the same units) between the tables being matched, and in geometrically sensitive cases such as matching on the sky, it's important that they are the units expected by the matcher as well. To determine what those units are, either consult the roster below, or run the following command:

```
stilts tmatch2 help=matcher
```

which will tell you about all the known matchers and their associated `params`, `values*` and `tuning` parameters.

The following subsections list the basic `matcher` types and the requirements of their associated `params`, `values*` and `tuning` parameters. The units of the required values are given where significant.

7.1.1 `sky`: Sky Matching

```
matcher=sky values*='<ra/degrees> <dec/degrees>'
            params='<max-error/arcsec>'
            tuning='<healpix-k>'
```

`values*`:

- `ra/degrees`: Right Ascension
- `dec/degrees`: Declination

`params`:

- `max-error/arcsec`: Maximum separation along a great circle

`tuning`:

- `healpix-k`: Controls sky pixel size. Legal range 0 - 29. 0 is 60deg, 20 is 0.2".

The `sky` matcher compares positions on the celestial sphere with a fixed error radius. Rows are considered to match when the two (`ra`, `dec`) positions are within `max-error` arcseconds of each other along a great circle.

In fact this matching is not restricted to equatorial coordinates - the `ra` and `dec` parameters may represent any longitude-like and latitude-like coordinates in degrees, since the spherical geometry for the matching is unchanged under such transformations.

7.1.2 `skyerr`: Sky Matching with Per-Object Errors

```
matcher=skyerr values*='<ra/degrees> <dec/degrees> <error/arcsec>'
               params='<scale/arcsec>'
               tuning='<healpix-k>'
```

`values*`:

- `ra/degrees`: Right Ascension
- `dec/degrees`: Declination
- `error/arcsec`: Per-object error radius along a great circle

`params`:

- `scale/arcsec`: Rough average of per-object error distance; just used for tuning to set default pixel size

`tuning`:

- `healpix-k`: Controls sky pixel size. Legal range 0 - 29. 0 is 60deg, 20 is 0.2".

The `skyerr` matcher compares positions on the celestial sphere using error radii which can be different for each row. Rows are considered to match when the separation between the two `ra`, `dec` positions is no larger than the sum of the two per-row `error` values.

The `scale` parameter should be a rough average value of the error distances. It is used only to set a sensible default for `healpix-k` tuning parameter, and its value does not affect the result. If you set `healpix-k` directly, its value is ignored.

As with `sky` matching, other longitude/latitude coordinate pairs may be used in place of right ascension and declination.

Note: the semantics of this matcher have changed slightly at version 2.4 of STILTS. In earlier versions the single parameter was named `max-error` and provided an additional constraint on the maximum accepted separation between matched objects. For most uses, the old and new behaviours

are expected to give the same results, but in cases of difference, the new behaviour is more likely what you want.

7.1.3 `skyellipse`: Sky Matching of Elliptical Regions

```
matcher=skyellipse values*='<ra/degrees> <dec/degrees> <primary-radius/arcsec>
                           <secondary-radius/arcsec>
                           <position-angle/degrees>'
                  params='<scale/arcsec>'
                  tuning='<healpix-k>'
```

`values*`:

- `ra/degrees`: Right ascension of centre
- `dec/degrees`: Declination of centre
- `primary-radius/arcsec`: Length of ellipse semi-major axis
- `secondary-radius/arcsec`: Length of ellipse semi-minor axis
- `position-angle/degrees`: Position angle - measured from north pole to primary axis, in direction of positive RA

`params`:

- `scale/arcsec`: Rough average of ellipse major radius; just used for tuning to set default pixel size

`tuning`:

- `healpix-k`: Controls sky pixel size. Legal range 0 - 29. 0 is 60deg, 20 is 0.2".

The `skyellipse` matcher compares elliptical regions on the sky for overlap. Each row has to provide five values, giving the centre, the major and minor radii, and the position angle of an ellipse. Rows are considered to match if there is any overlap between the ellipses. The goodness of match is a normalised generalisation of the symmetrical case used by the `skyerr` matcher, in which the best possible match is two concentric ellipses, and the worst allowable match is when the circumferences just touch.

The calculations are approximate since in some cases they rely on projecting the ellipses onto a Cartesian tangent plane before evaluating the match, so for larger ellipses the criterion will be less exact. For objects the size of most observed stars or galaxies, this approximation is not expected to be problematic.

The `scale` parameter must be supplied, and should be a rough average value of the major radii. it is used only to set a sensible default for the `healpix-k` tuning parameter, and its value does not affect the result. If you set `healpix-k` directly, the value of `scale` is ignored.

7.1.4 `sky3d`: Spherical Polar Matching

```
matcher=sky3d values*='<ra/degrees> <dec/degrees> <distance>'
              params='<error/units of distance>'
              tuning='<bin-factor>'
```

`values*`:

- `ra/degrees`: Right Ascension
- `dec/degrees`: Declination
- `distance`: Distance from origin

params:

- `error/units of distance`: Maximum Cartesian separation for match

tuning:

- `bin-factor`: Scaling factor to adjust bin size; larger values mean larger bins

The `sky3d` matcher compares positions in the volume of the sky taking account of distance from the observer. The position in three-dimensional space is calculated for each row using the `ra`, `dec` and `distance` as spherical polar coordinates, where `distance` is the distance from the observer along the line of sight. Rows are considered to match when their positions in this space are within `error` units of each other. The units of `error` are the same as those of `distance`.

As with `sky` matching, other longitude/latitude coordinate pairs may be used in place of right ascension and declination.

7.1.5 `exact`: Exact Matching

```
matcher=exact values*='<matched-value>'
```

values*:

- `matched-value`: Value for exact match

The `exact` matcher compares arbitrary key values for exact equality. Rows are considered to match only if the values in their `matched-value` columns are exactly the same. These values can be strings, numbers, or anything else. A blank value never matches, not even with another blank one. Since the `params` parameter holds no values, it does not have to be specified. Note that the values must also be of the same type, so for instance a Long (64-bit) integer value will not match an Integer (32-bit) value.

7.1.6 `1d`, `2d`, ...: Isotropic Cartesian Matching

```
matcher=1d values*='<x>'
          params='<error>'
          tuning='<bin-factor>'
```

values*:

- `x`: Cartesian co-ordinate #1

params:

- `error`: Maximum Cartesian separation for match

tuning:

- `bin-factor`: Scaling factor to adjust bin size; larger values mean larger bins

```
matcher=2d values*='<x> <y>'
          params='<error>'
```

```
tuning='<bin-factor>'
```

values*:

- x: Cartesian co-ordinate #1
- y: Cartesian co-ordinate #2

params:

- error: Maximum Cartesian separation for match

tuning:

- bin-factor: Scaling factor to adjust bin size; larger values mean larger bins

The 1d matcher compares positions in 1-dimensional Cartesian space. Rows are considered to match if their x column values differ by no more than error.

The 2d matcher compares postions in 2-dimensional Cartesian space. Rows are considered to match if the difference in their (x,y) positions reckoned using Pythagoras is less than error.

Matching in any number of Cartesian dimensions can be done by extending this syntax in the obvious way.

7.1.7 2d_anisotropic, ...: Anisotropic Cartesian Matching

```
matcher=2d_anisotropic values*='<x> <y>'
                       params='<error-in-x> <error-in-y>'
                       tuning='<bin-factor>'
```

values*:

- x: Cartesian co-ordinate #1
- y: Cartesian co-ordinate #2

params:

- error-in-x: Axis length of error ellipse in Cartesian co-ordinate #1 direction
- error-in-y: Axis length of error ellipse in Cartesian co-ordinate #2 direction

tuning:

- bin-factor: Scaling factor to adjust bin size; larger values mean larger bins

The 2d_anisotropic matcher compares positions in 2-dimensional Cartesian space using an anisotropic metric. Rows are considered to match if their (x,y) positions fall within an error ellipse with axis lengths error-in-x, error-in-y of each other. This kind of match will typically be used for non-'spatial' spaces, for instance (magnitude,redshift) space, in which the metrics along different axes are not related to each other.

Matching in any number of dimensions of Cartesian space using an anisotropic metric can be done by extending this syntax in the obvious way.

7.1.8 2d_cuboid, ...: Cuboid Cartesian Matching

```
matcher=2d_cuboid values*='<x> <y>'
                  params='<error-in-x> <error-in-y>'
                  tuning='<bin-factor>'
```

values*:

- x: Cartesian co-ordinate #1
- y: Cartesian co-ordinate #2

params:

- error-in-x: Half length of cuboid in Cartesian co-ordinate #1 direction
- error-in-y: Half length of cuboid in Cartesian co-ordinate #2 direction

tuning:

- bin-factor: Scaling factor to adjust bin size; larger values mean larger bins

The 2d_cuboid matcher compares positions in 2-dimensional Cartesian space in cuboidal cells. Rows are considered to match if their (x,y) positions fall within an error cuboid with half-axis lengths error-in-x, error-in-y of each other. This kind of match is suitable for grouping items into pixels, though it's not a very efficient way of doing that.

Matching in any number of dimensions using N-dimensional hyper-cuboids can be done by extending this syntax in the obvious way.

7.1.9 1d_err, 2d_err, ...: Cartesian Matching with Per-Object Errors

```
matcher=2d_err values*='<x> <y> <error>'
               params='<scale>'
               tuning='<bin-factor>'
```

values*:

- x: Cartesian co-ordinate #1
- y: Cartesian co-ordinate #2
- error: Per-object error radius

params:

- scale: Rough average of per-object error distance; just used for tuning in conjunction with bin factor

tuning:

- bin-factor: Scaling factor to adjust bin size; larger values mean larger bins

The 1d_err, 2d_err, ... matchers compare positions in N-dimensional Cartesian space like the 1d, 2d matchers described in Section 7.1.6, except that the match radius can be different for each row. Rows are considered to match when the separation reckoned by Pythagoras between the x, y, ... positions is no larger than the sum of the two per-row error values. Matching in any number of Cartesian dimensions can be done by extending this syntax in the obvious way.

The scale parameter must be supplied, and should be approximately the characteristic size of the per-object error values. In conjunction with the bin-factor tuning parameter its value affects the

performance of the match, but not the result.

7.1.10 `2d_ellipse`: Cartesian Matching of Elliptical Regions

```
matcher=2d_ellipse values*='<x> <y> <primary-radius> <secondary-radius>
                           <orientation-angle/degrees>'
                   params='<scale>'
                   tuning='<bin-factor>'
```

`values*`:

- `x`: X coordinate of centre
- `y`: Y coordinate of centre
- `primary-radius`: Length of ellipse semi-major axis
- `secondary-radius`: Length of ellipse semi-minor axis
- `orientation-angle/degrees`: Angle from X axis towards Y axis of semi-major axis

`params`:

- `scale`: Rough average of per-object error distance; just used for tuning in conjunction with bin factor

`tuning`:

- `bin-factor`: Scaling factor to adjust bin size; larger values mean larger bins

The `2d_ellipse` matcher compares elliptical regions in a 2d plane for overlap. Each row has to specify five values, giving the centre, the major and minor radii, and the orientation angle of an ellipse. Rows are considered to match if there is any overlap between the ellipses. The goodness of match is a normalised generalisation of the symmetrical case used by the isotropic matcher, in which the best possible match is two concentric ellipses, and the worst allowable match is when the circumferences just touch.

Note the orientation angle is measured anticlockwise from the horizontal, unlike the position angle used by the `skyellipse` matcher.

The `scale` parameter must be supplied, and should be approximately the characteristic size of the per-object major radius. In conjunction with the `bin-factor` tuning parameter its value affects the performance of the match, but not the result.

7.1.11 Custom Matchers

For advanced users, it is possible to supply the name of a class on the classpath which implements the `uk.ac.starlink.table.join.MatchEngine` interface and which has a no-arg constructor. This allows java programmers to write their own matchers using any match criteria and binning algorithms they choose.

7.1.12 Matcher Combinations

In addition to the matching criteria listed in the previous subsections, you can build your own by combining any of these. To do this, take the two (or more) matchers that you want to use, and separate their names with a "+" character. The `values*` parameters of the combined matcher should then hold the concatenation of the `values*` entries of the constituent matchers, and the same for the `params` parameter.

So for instance the matcher "sky+1d" could be used with the following syntax:

```
matcher=sky+1d values*='<ra/degrees> <dec/degrees> <x>'
               params='<max-error/arcsec> <error>'
               tuning='<healpix-k> <bin-factor>'
```

values*:

- ra/degrees: Right Ascension
- dec/degrees: Declination
- x: Cartesian co-ordinate #1

params:

- max-error/arcsec: Maximum separation along a great circle
- error: Maximum Cartesian separation for match

tuning:

- healpix-k: Controls sky pixel size. Legal range 0 - 29. 0 is 60deg, 20 is 0.2".
- bin-factor: Scaling factor to adjust bin size; larger values mean larger bins

This would compare positions on the sky with an additional scalar constraint. Rows are considered to match if *both* their ra, dec positions are within max-error arcseconds of each other along a great circle (as for matcher=sky) *and* their x values differ by no more than error (as for matcher=1d).

This example might be used for instance to identify objects from two catalogues which are within a couple of arcseconds and also 0.5 blue magnitudes of each other. Rolling your own matchers in this way can give you quite flexible match constraints.

When identifying the closest match (e.g. find=best1 in tmatch2) the "distance" measure is obtained by scaling the distances from each of the constituent matchers and adding these scaled distances in quadrature, so that each element of the matcher has approximately equal weight. Scaling is generally done using the maximum permissible match radius (or equivalent), so the distance measure looks something like $d = sqrt([d_A/max(d_A)]^2 + [d_B/max(d_B)]^2)$. However the details are a bit dependent on which matchers you are combining.

Note that in STILTS v3.0-9 and earlier, a linear unscaled distance measure was used here instead, which did not give very meaningful Best match results.

7.2 Multi-Object Matches

The generic matching in STILTS is determined by specified match criteria, as described in Section 7.1. These criteria give conditions for whether two items (table rows) count as matched with each other. In the case of a pair match, as provided by tmatch2, it is clear how this is to be interpreted.

However, some of the matching tasks (tmatchn in group mode and tmatch1) search for match groups which may have more than two members. This section explains precisely how STILTS applies the pair-wise matching criteria it is given to identifying multi-object groups.

In a multi-object match context, the matcher identifies a matched group as the largest possible group of objects in which each is linked by a pair match to *any* other object in the group - it is a group of "friends of friends". Formally, the set of matched groups is a set of disjoint graphs whose nodes are input table rows and whose edges are successful pair matches, where no successful pair match exists between nodes in different elements of that set. Thus the set has a minimal number of elements, and each of its elements is a matched group of maximal size. The important point to note is that for any particular pair in a matched group, there is no guarantee that the two objects match

each other, only that you can hop from one to the other via pairs which do match.

So in the case of a multi-object sky match on a field which is very crowded compared to the specified error radius, it is quite possible for *all* the objects in the input table(s) to end up as part of the same large matching group. Results at or near this percolation threshold are (a) probably not useful and (b) likely to take a long time to run. Some care should therefore be exercised when specifying match criteria in multi-object match contexts.

8 Plotting

As of version 3.0 (October 2014), STILTS offers plotting commands corresponding to the new-style plots in version 4 of the TOPCAT application. The commands are currently:

- `plot2plane` (Appendix B.10): Draws a plane plot
- `plot2sky` (Appendix B.11): Draws a sky plot
- `plot2cube` (Appendix B.12): Draws a cube plot
- `plot2sphere` (Appendix B.13): Draws a sphere plot
- `plot2time` (Appendix B.14): Draws a time plot

(In previous versions the less capable commands `plot2d`, `plot3d` and `plothist` were available - these are now deprecated, but described in Section 9).

These commands all have a similar structure. The *plot surface*, or geometry of the plot, is defined by which command you use (for instance, if you want to plot longitude/latitude data on the celestial sphere, use `plot2sky`). Content is added to the plot by specifying zero or more *plot layers*, as described in Section 8.3 below. Section 8.4 describes the shading modes which affect how colouring is performed for some of the layer types. Once a plot has been specified, it can be displayed on the screen or exported in some way according to a selected output mode (Section 8.5) and perhaps export format (Section 8.6). Plots displayed to the screen are by default "live" - they can be resized and navigated around (pan, zoom, rotate, ...) using the mouse in the same way as in a TOPCAT window.

These commands allow you to make all the plots that can be produced with TOPCAT, in some cases with more flexibility in configuration. Unlike TOPCAT, the size of table you can plot is not limited by the size of table you can load into the application. In most cases, STILTS will generate plots from arbitrarily large data sets with fixed (and modest) memory requirements. Performance is of course highly dependent on the details of the plot, but for instance an all-sky density plot for 2 billion points can be produced in the order of 30 minutes.

8.1 Plot Parameters

The plotting commands offer a great deal of control over what is plotted and how it is represented, and thus unavoidably have lots of parameters. When looking at the command documentation in Appendix B the Usage sections may look rather daunting. However, the discussion below and the Examples sections should help. Generating a simple plot is straightforward and can be done with only four or five parameters; if you want to represent more complicated data or have specific preferences for appearance then you can consult the documentation for the additional options.

As a simple example, if a file "cat.fits" contains the columns RMAG and BMAG for red and blue magnitudes, you can draw a two-dimensional colour-magnitude scatter plot with the command:

```
stilts plot2plane layer_1=mark in_1=cat.fits x_1=BMAG-RMAG y_1=BMAG
```

Since an output file is not specified, the plot is shown in a window on the screen. This plot window is "live" - you can resize the window, or pan and zoom around it using the same mouse controls as in TOPCAT. To send the output to a PNG file, do instead:

```
stilts plot2plane layer_1=mark in_1=cat.fits x_1=BMAG-RMAG y_1=BMAG out=fig.png
```

We can adjust the plot by inverting the Y axis so it increases downwards instead of upwards:

```
stilts plot2plane
           yflip=true
           layer_1=mark in_1=cat.fits x_1=BMAG-RMAG y_1=BMAG
```

The parameters of the plot now fall into two groups. Global parameters, without suffixes, make global adjustments to the plot. In this example `yflip=true` inverts the Y axis. **Layer parameters**, with suffixes, are introduced by a `layer` parameter and grouped together by a given suffix. Each layer group defines a plot layer with content to be drawn on the plot surface. In this case the layer is of type `mark` (draw markers) and the suffix is "`_1`". Global and Layer parameters are described separately in the following subsections.

8.1.1 Global Parameters

The global plot parameters are documented in the usage sections of the various plot commands (e.g. Appendix B.10.1). They deal with things like positioning the plot axes, fixing the data bounds, selecting font types and sizes, adjusting grids and tickmarks, configuring how interactive navigation works, managing data storage, and so on. They are all optional, since they all have sensible defaults, for instance data bounds will be determined from the supplied data if they are not given explicitly.

8.1.2 Layer Parameters

The layer parameters come in groups, each specifying the details of one plot layer. Each layer type has its own list of parameters. A plot layer is introduced on the command line with a parameter of the form

```
layer<suffix>=<layer-type>
```

and any other parameters with the same `<suffix>` are considered to apply to the same layer. In the basic example we considered:

```
stilts plot2plane layer_1=mark in_1=cat.fits x_1=BMAG-RMAG y_1=BMAG
```

the suffix is "`_1`" and the layer type associated with it is `mark` (plotting markers to make a scatter plot). The different layer types are documented in Section 8.3, and each has its own set of parameters, some of which are mandatory and some which are optional with sensible defaults. In the documentation, the suffix is represented as "`N`". For instance the `mark` layer type requires you to specify an input table (`inN`) and point positions (`xN` and `yN`). Since the suffix we have used in the example for the `layerN` parameter is "`_1`", we have written `in_1`, `x_1` and `y_1`. The `mark` layer has some optional style parameters as well, so we could adjust the plot's appearance by adding `shape_1=cross size_1=4 color_1=blue`.

You can have as many layers as you like (even none), so we could overplot two datasets from different input files like this:

```
stilts plot2plane
    layer_1=mark in_1=cat1.fits x_1=BMAG-RMAG   y_1=BMAG  color_1=magenta size_1=5
    layer_2=mark in_2=cat2.fits x_2=mag_b-mag_r y_2=mag_b color_2=cyan    size_2=5
```

We have assigned different colours to the different layers and boosted the marker size to 5 pixels.

As a convenience, if the same value is used for all the layers, you can omit the suffix. So to avoid having to specify the same markers size for both layers, you can write instead:

```
stilts plot2plane
    size=5
    layer_1=mark in_1=cat1.fits x_1=BMAG-RMAG   y_1=BMAG  color_1=magenta
    layer_2=mark in_2=cat2.fits x_2=mag_b-mag_r y_2=mag_b color_2=teal
```

Although the `size` parameter no longer has an explicit suffix, it's still a layer parameter, it just applies to multiple layers. This shorthand works for all layer parameters. Here is another example which also shows how you can use the `icmdN` parameter to pre-process input data prior to performing the plot. Here, we make two different selections of the input rows to plot two different data sets.

```
stilts plot2plane
       in=cat.fits x=BMAG-RMAG y=BMAG
       layer_1=mark icmd_1='select vel<1000'  color_1=blue
       layer_2=mark icmd_2='select vel>=1000' color_2=red
```

The input tables and data values are the same for both datasets, so we can just supply the parameters `in`, `x` and `y`, rather than `in_1`, `in_2` etc.

Any string can be used as a suffix, including the empty string (though an empty string can cause confusion if there are multiple layers). The suffixing is also slightly more sophisticated than described above; to find parameters relating to a layer with a given suffix, the parameter looks first using the whole suffix, and strips single characters off it until it has none left. So if a layer is introduced with the parameter `layer_ab`, you can give the marker shape using any of the parameters `shape_ab`, `shape_a`, `shape_` or `shape`. If more than one of these is present, the first one in that list will be used (the order in which they appear on the command line is not significant). This can be used to group sets of layers.

By default, if multiple layers are specified, they are plotted in the order in which the introducing `layerN` parameters appear on the command line. This may be relevant, since layers plotted later sometimes obscure ones plotted earlier. You can alter the order of plotting with the `seq` (global) parameter, which is a comma-separated list of layer suffixes giving the sequence in which layers should be plotted. So adding "`seq=_2,_1`" would cause layer _2 to be plotted before layer _1, instead of the other way round.

By default, if more than one layer is plotted, a legend will appear labelling the datasets. The dataset labels appearing in the legend are by default the layer suffixes specified on the command line. However, the labels can be given explicitly with the `legendN` parameter, so for instance in the example above `leglabel_1=Slow leglabel_2=Fast` would adjust the legend accordingly. Legend appearance and positioning can be adjusted by various `leg*` global parameters.

8.1.3 Animation

The plotting commands can be used to produce animations. This is done by supplying an *animation control table* using the `animate` parameter (which has associated `afmt` and `acmd` parameters for specifying its file format and applying filters). One output image is produced for each row of the control table. The columns of the table have names which correspond to plot command parameters, and for each row, the basic plot command is executed with the parameters on the command line supplied or replaced by those from the table. This is most commonly used for providing a movie of the kind of navigation you can do interactively with the mouse, but other applications are possible.

For instance, given the following animation control table with the name "bounds.txt", in ASCII format:

```
#  xmax  ymax
    4.0   2.0
    3.0   1.5
    2.0   1.0
    1.0   0.5
```

then this command:

```
stilts plot2plane xmin=0 ymin=0
                  layer_1=mark in_1=gums_smc.fits x_1=ag y_1=av
                  animate=bounds.txt afmt=ascii
```

would produce a 4-frame animation zooming in towards the origin.

If output is to the screen (`omode=swing`, the default) the animation can be seen directly. If it is to an output file (`omode=out`) then a number of output files is written with sequence numbers, so adding

the parameter "out=x.png" to the above command would produce 4 files, x-1.png, x-2.png, x-3.png and x-4.png. Padding zeros are used to keep the files in alphanumeric sequence, so for instance in a 500-frame animation the first one would be named x-001.png. STILTS does not actually turn these files into a single animated output file, but you can use other tools to do this, for instance using ImageMagick:

```
convert x-*.png xmovie.gif
```

or ffmpeg:

```
ffmpeg -i 'x-%03d.png' -framerate 15 -pix_fmt yuv420p xmovie.webm
```

You can create the animation control table any way you like, but you may find the tloop command convenient. For instance the above table can be written like this:

```
stilts tloop xmax 4 0 -1 ocmd='addcol ymax xmax*0.5' ofmt=ascii out=bounds.txt
```

Alternatively, you can generate a table like this inline using the loop scheme. A common requirement is to produce an animation of rotating a 3-d plot, here's an example of how to do that:

```
stilts plot2sphere layer_1=mark in_1=hip_main.fits lon_1=radeg lat_1=dedeg r_1=plx \
                   animate=:loop:15,375,2 acmd='colmeta -name phi $1'
```

The phi parameter controls the angle from which the 3D plot is viewed, and here it is incremented by 2 degrees for each frame. The same thing would work for plot2cube as well as plot2sphere.

Note that producing animations in this way is usually much more efficient than writing a shell script which invokes STILTS multiple times. The plot commands also employ multi-threading when animating to output files, so should make efficient use of multi-core machines (though currently animations to the screen are not multi-threaded).

8.2 Surface Types

The different plot2* commands correspond to different plot surface geometries. The different commands come with their own specific axis configuration parameters. Some of the plot layer types are specific to certain surface types. When supplying data from input tables to plot layers, the coordinate values you need to supply (and hence the corresponding parameter names) are determined not by the layer type, but by the surface type. For instance, point positions for layer N on a 2-d Cartesian surface (plot2plane command) are given using parameters xN and yN, but when plotting to the celestial sphere (plot2sky command) you supply lonN and latN).

The following list summarises the available surface types and their corresponding positional coordinates.

Plane (plot2plane)
: 2-dimensional Cartesian axes. Positional coordinates are supplied as x, y pairs. Note that this command can also be used to draw histograms.

Sky (plot2sky)
: Celestial sphere. Positional coordinates are supplied as lon, lat pairs, giving longitude and latitude in decimal degrees. A number of different projections are available, and conversion between different celestial coordinate systems can also be performed. You could use it for other spherical coordinate systems too (like the surface of a planet).

Cube (plot2cube)
: 3-dimensional Cartesian axes. Positional coordinates are supplied as x, y, z triples.

Sphere (plot2sphere)
: 3-dimensional isotropic space with spherical polar coordinates. Positional coordinates are

supplied as `lon`, `lat`, `r` triples, giving longitude and latitude in decimal degrees, and radius in an arbitrary unit. The plotting surface (space) is similar to Cube, except that the unit distance is always the same in all three directions.

Time (`plot2time`)
2-dimensional axes, but the horizontal axis represents time. The axis may be labelled in various ways (ISO-8601 dates, decimal year, MJD etc). Positional coordinates are supplied as `t`, `y` pairs. Time can be represented in input data in various ways; if sufficient metadata is provided in the input format the epoch can be determined automatically, otherwise it may be necessary to specify the time representation being used.

8.3 Layer Types

The different plot layers and how to configure them with parameters is given in the following subsections. The layers which may be plotted on a particular surface depend on the plot geometry, so not all of these are available for every plot command.

8.3.1 `mark`

Plots a marker of fixed size and shape at each position.

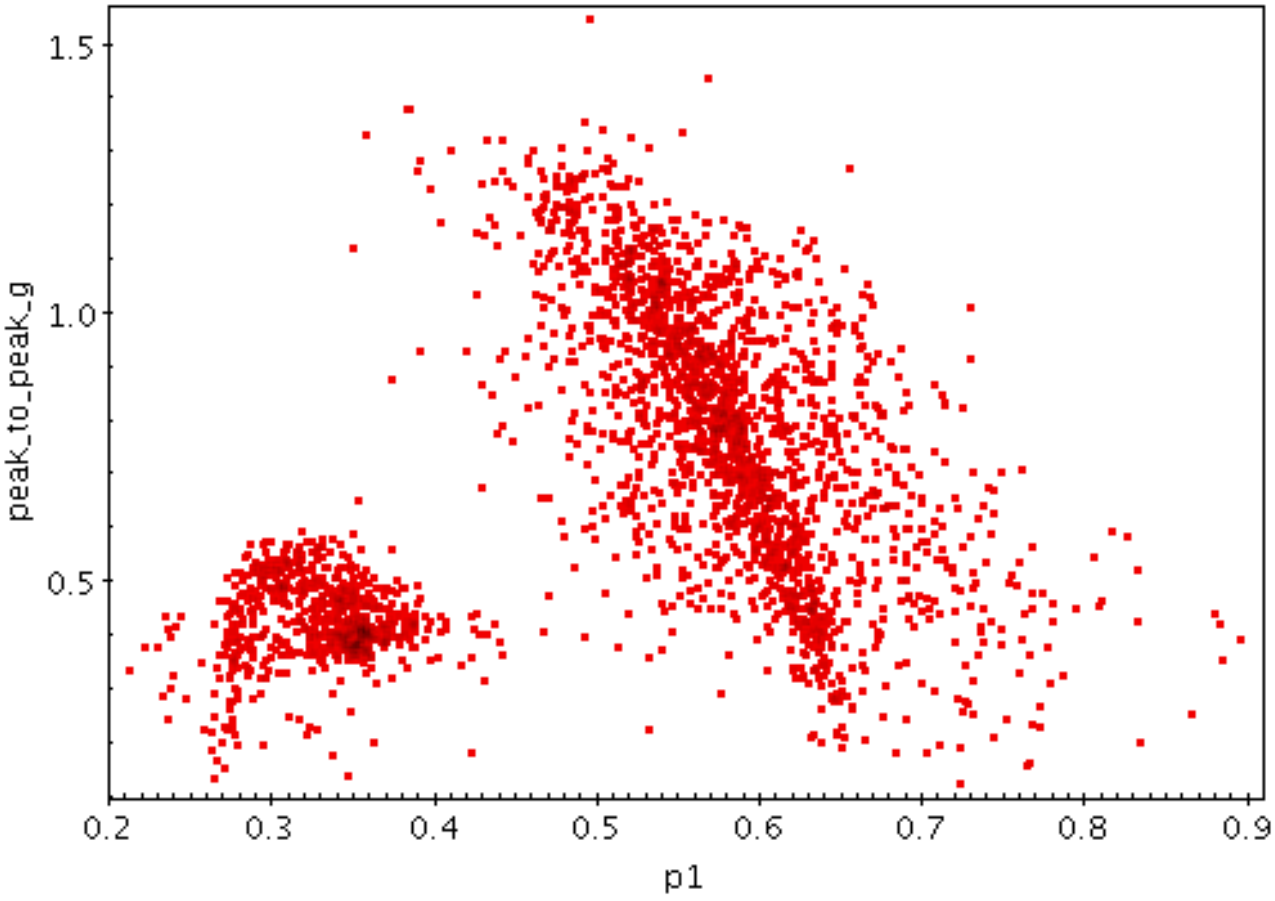
Usage Overview:

```
layerN=mark shapeN=filled_circle|open_circle|... sizeN=<pixels>
            shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>
            <pos-coord-paramsN> inN=<table> ifmtN=<in-format>
            istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:
The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2plane layer1=mark in1=rrlyrae.fits x1=p1 y1=peak_to_peak_g
```

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given

explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.

- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`shapeN = filled_circle|open_circle|...` ***(MarkerShape)***

Sets the shape of markers that are plotted at each position of the scatter plot.

The available options are:

- `filled_circle`
- `open_circle`
- `cross`
- `x`
- `open_square`
- `open_diamond`
- `open_triangle_up`
- `open_triangle_down`
- `fat_circle`
- `fat_cross`
- `fat_x`
- `fat_square`
- `fat_diamond`
- `fat_triangle_up`
- `fat_triangle_up`
- `filled_square`

- `filled_diamond`
- `filled_triangle_up`
- `filled_triangle_down`

[Default: `filled_circle`]

`sizeN = <pixels>` ***(Integer)***
Size of the scatter plot markers. The unit is pixels, in most cases the marker is approximately twice the size of the supplied value.

[Default: `1`]

8.3.2 `size`

Plots a marker of fixed shape but variable size at each position. The size is determined by an additional input data value.

The actual size of the markers depends on the setting of the `autoscale` parameter. If autoscaling is off, then the basic size of each marker is the input data value in units of pixels. If autoscaling is on, then the data values are gathered for all the currently visible points, and a scaling factor is applied so that the largest ones will be a sensible size (a few tens of pixels). This basic size can be further adjusted with the `scale` factor.

Currently data values of zero always correspond to marker size of zero, negative data values are not represented, and the mapping is linear. An absolute maximum of 100 pixels is also imposed on marker sizes. Other options may be introduced in future.

Note: for marker sizes that correspond to data values in data coordinates, you may find Error plotting more appropriate.

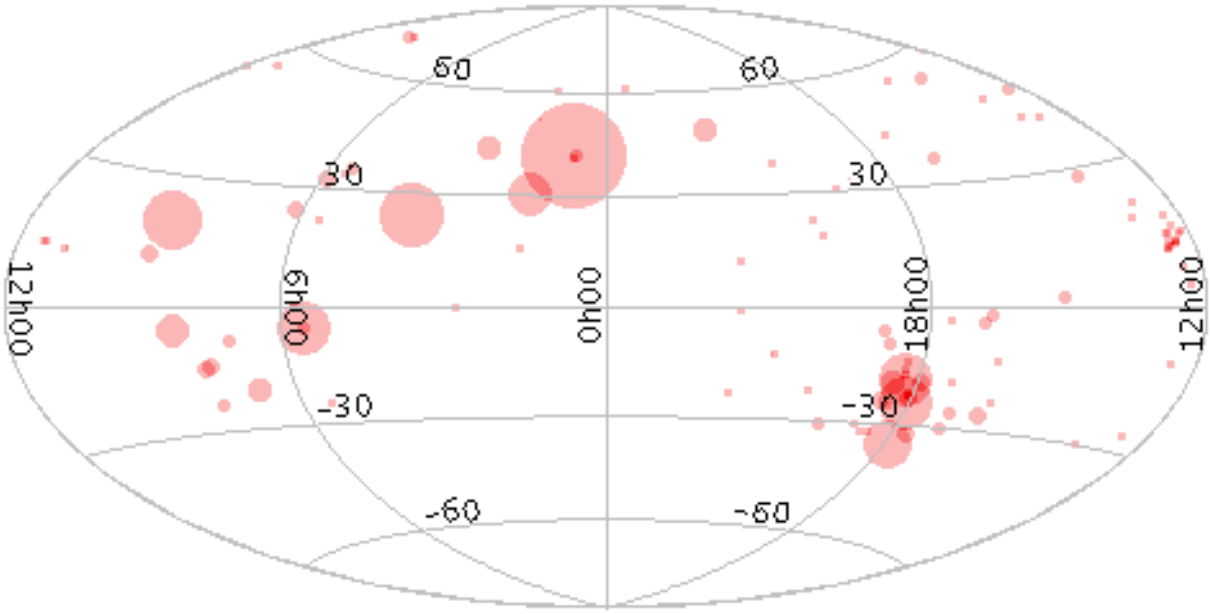
Usage Overview:

```
layerN=size shapeN=filled_circle|open_circle|... scaleN=<factor>
            autoscaleN=true|false
            shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>
            <pos-coord-paramsN> sizeN=<num-expr> inN=<table>
            ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:
The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2sky projection=aitoff xpix=500 ypix=250
                layer1=size in1=messier.xml shading1=transparent lon1=RA lat1=DEC size1=Rad
```

`autoscaleN = true|false` ***(Boolean)***

Determines whether the basic size of variable sized markers is automatically scaled to have a sensible size. If true, then the sizes of all the plotted markers are examined, and some dynamically calculated factor is applied to them all to make them a sensible size (by default, the largest ones will be a few tens of pixels). If false, the sizes will be the actual input values in units of pixels.

If auto-scaling is off, then markers will keep exactly the same screen size during pan and zoom operations; if it's on, then the visible sizes will change according to what other points are currently plotted.

Marker size is also affected by the `scale` parameter.

[Default: `true`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`scaleN = <factor>` ***(Double)***

Scales the size of variable-sized markers. The default is 1, smaller or larger values multiply the visible sizes accordingly.

[Default: `1`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`shapeN = filled_circle|open_circle|...` ***(MarkerShape)***

Sets the shape of markers that are plotted at each position of the scatter plot.

The available options are:

- `filled_circle`
- `open_circle`
- `cross`
- `x`

- open_square
- open_diamond
- open_triangle_up
- open_triangle_down
- fat_circle
- fat_cross
- fat_x
- fat_square
- fat_diamond
- fat_triangle_up
- fat_triangle_up
- filled_square
- filled_diamond
- filled_triangle_up
- filled_triangle_down

[Default: filled_circle]

sizeN = <num-expr> ***(String)***

Size to draw each sized marker. Units are pixels unless auto-scaling is in effect, in which case units are arbitrary. The plotted size is also affected by the scale value.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.3 sizexy

Plots a shaped marker with variable horizontal and vertical extents at each position. The X and Y dimensions are determined by two additional input data values.

The actual size of the markers depends on the setting of the autoscale parameter. If autoscaling is off, the basic dimensions of each marker are given by the input data values in units of pixels. If autoscaling is on, the data values are gathered for all the currently visible points, and scaling factors are applied so that the largest ones will be a sensible size (a few tens of pixels). This autoscaling happens independently for the X and Y directions. The basic sizes can be further adjusted with the scale factor.

Currently data values of zero always correspond to marker dimension of zero, negative data values are not represented, and the mapping is linear. An absolute maximum of 100 pixels is also imposed on marker sizes. Other options may be introduced in future.

Note: for marker sizes that correspond to data values in data coordinates, you may find Error plotting more appropriate.

Usage Overview:

```
layerN=sizexy shapeN=open_rectangle|open_triangle|... thickN=<int-value>
              scaleN=<factor> autoscaleN=true|false
              shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-params
              <pos-coord-paramsN> xsizeN=<num-expr> ysizeN=<num-expr>
              inN=<table> ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

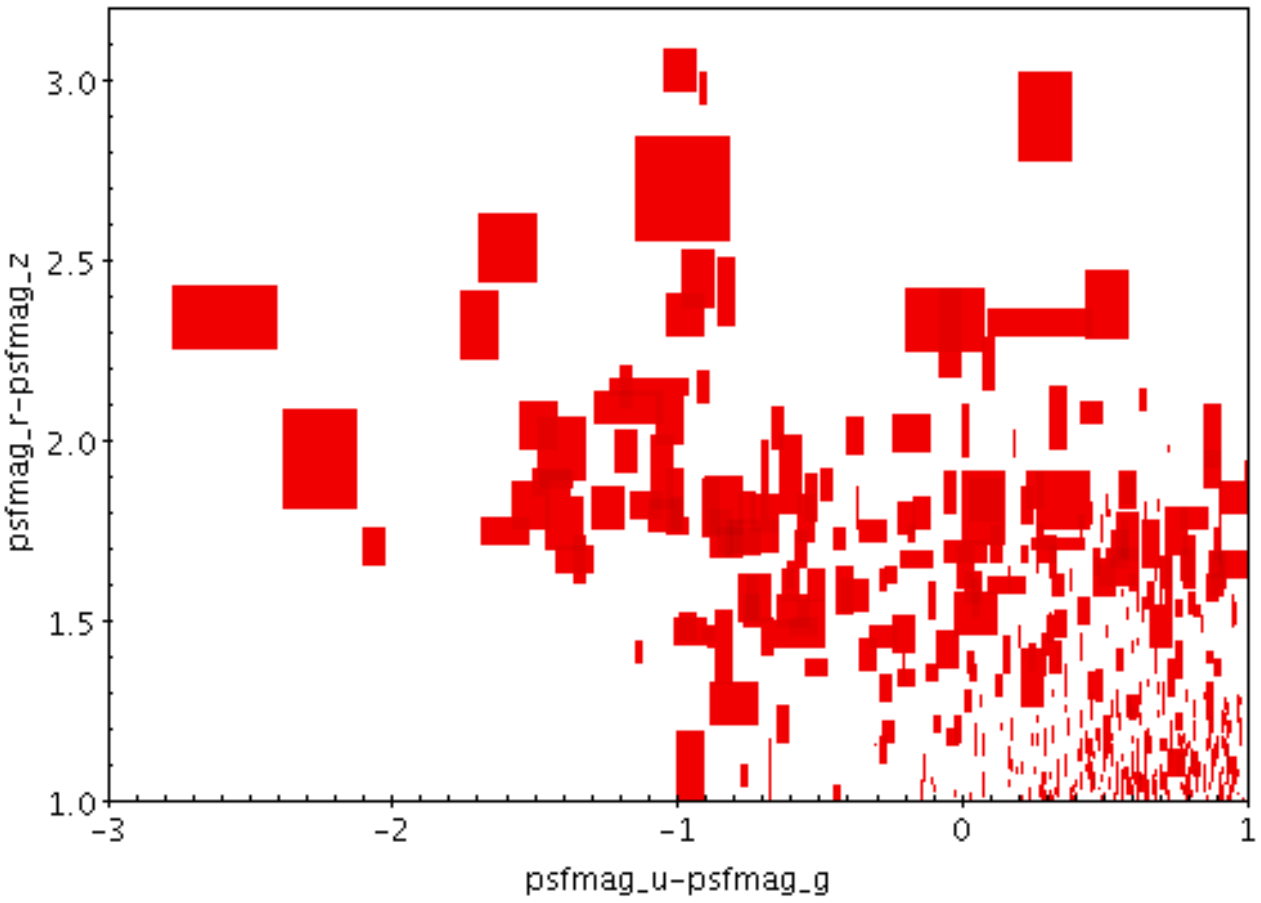
All the parameters listed here affect only the relevant layer, identified by the suffix N.

Positional Coordinate Parameters:

The positional coordinates <pos-coord-paramsN> give a position for each row of the input

table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2plane layer1=sizexy in1=dr5qso.fits shape1=filled_rectangle
                  x1=psfmag_u-psfmag_g y1=psfmag_r-psfmag_z xsize1=exp(psfmag_g) ysize1=exp
                  xmin=-3 xmax=1 ymin=1 ymax=3.2
```

`autoscaleN = true|false` ***(Boolean)***

Determines whether the basic size of variable sized markers is automatically scaled to have a sensible size. If true, then the sizes of all the plotted markers are examined, and some dynamically calculated factor is applied to them all to make them a sensible size (by default, the largest ones will be a few tens of pixels). If false, the sizes will be the actual input values in units of pixels.

If auto-scaling is off, then markers will keep exactly the same screen size during pan and zoom operations; if it's on, then the visible sizes will change according to what other points are currently plotted.

Marker size is also affected by the `scale` parameter.

[Default: `true`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter

can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`scaleN = <factor>` ***(Double)***

Scales the size of variable-sized markers. The default is 1, smaller or larger values multiply the visible sizes accordingly.

[Default: `1`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`

- `density` (Section 8.4.5)
- `aux` (Section 8.4.6)
- `weighted` (Section 8.4.7)

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`shapeN = open_rectangle|open_triangle|...` ***(BasicXYShape)***

The available options are:

- `open_rectangle`
- `open_triangle`
- `open_triangle_down`
- `open_diamond`
- `open_ellipse`
- `filled_rectangle`
- `filled_triangle`
- `filled_triangle_down`
- `filled_diamond`
- `filled_ellipse`

[Default: `open_rectangle`]

`thickN = <int-value>` ***(Integer)***

Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

`xsizeN = <num-expr>` ***(String)***

Horizontal extent of each marker. Units are pixels unless auto-scaling is in effect, in which case units are arbitrary.

The value is a numeric algebraic expression based on column names as described in Section 10.

`ysizeN = <num-expr>` ***(String)***

Vertical extent of each marker. Units are pixels unless auto-scaling is in effect, in which case units are arbitrary.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.4 `xyvector`

Plots directed lines from the data position given delta values for the coordinates. The plotted markers are typically little arrows, but there are other options.

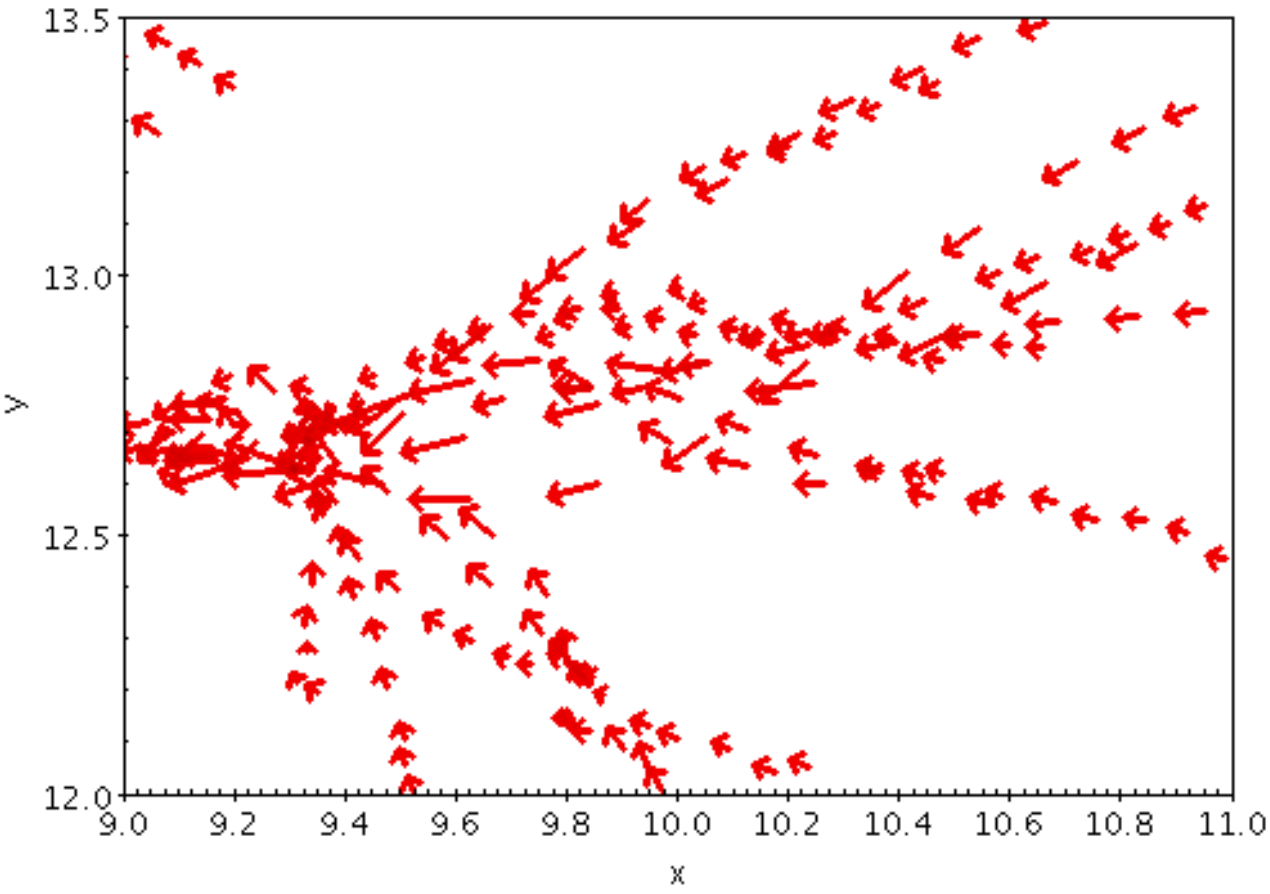
In some cases the supplied data values give the actual extents in data coordinates for the plotted vectors but sometimes the data is on a different scale or in different units to the positional coordinates. As a convenience for this case, the plotter can optionally scale the magnitudes of all the vectors to make them a reasonable size on the plot, so by default the largest ones are a few tens of pixels long. This auto-scaling is turned off by default, but it can be activated with the `autoscale` option. Whether autoscaling is on or off, the `scale` option can be used to apply a fixed scaling factor.

Usage Overview:

```
layerN=xyvector arrowN=small_arrow|medium_arrow|... thickN=<int-value>
                scaleN=<factor> autoscaleN=true|false
                shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-para
                xN=<num-expr> yN=<num-expr> xdeltaN=<num-expr>
                ydeltaN=<num-expr> inN=<table> ifmtN=<in-format>
                istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane layer1=xyvector in1=gavo_g2.fits
                  x1=x y1=y xdelta1=velX ydelta1=velY autoscale1=true thick1=1
                  xmin=9 xmax=11 ymin=12 ymax=13.5
```

`arrowN = small_arrow|medium_arrow|...` ***(MultiPointShape)***

How arrows are represented.

The available options are:

- `small_arrow`
- `medium_arrow`
- `large_arrow`
- `small_open_dart`
- `medium_open_dart`
- `large_open_dart`
- `small_filled_dart`
- `medium_filled_dart`
- `large_filled_dart`
- `lines`

- `capped_lines`

[Default: `small_arrow`]

`autoscaleN = true|false` ***(Boolean)***

Determines whether the default size of variable-sized markers like vectors and ellipses are automatically scaled to have a sensible size. If true, then the sizes of all the plotted markers are examined, and some dynamically calculated factor is applied to them all to make them a sensible size (by default, the largest ones will be a few tens of pixels). If false, the sizes will be the actual input values interpreted in data coordinates.

If auto-scaling is on, then markers will keep approximately the same screen size during zoom operations; if it's off, they will keep the same size in data coordinates.

Marker size is also affected by the `scale` parameter.

[Default: `false`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be

streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`scaleN = <factor>` ***(Double)***
Affects the size of variable-sized markers like vectors and ellipses. The default value is 1, smaller or larger values multiply the visible sizes accordingly.

[Default: `1`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***
Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`thickN = <int-value>` ***(Integer)***
Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

`xN = <num-expr>` ***(String)***
Horizontal coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xdeltaN = <num-expr>` ***(String)***
Vector component in the X direction.

The value is a numeric algebraic expression based on column names as described in Section 10.

`yN = <num-expr>` ***(String)***
Vertical coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`ydeltaN = <num-expr>` ***(String)***
Vertical component in the Y direction.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.5 `xyerror`

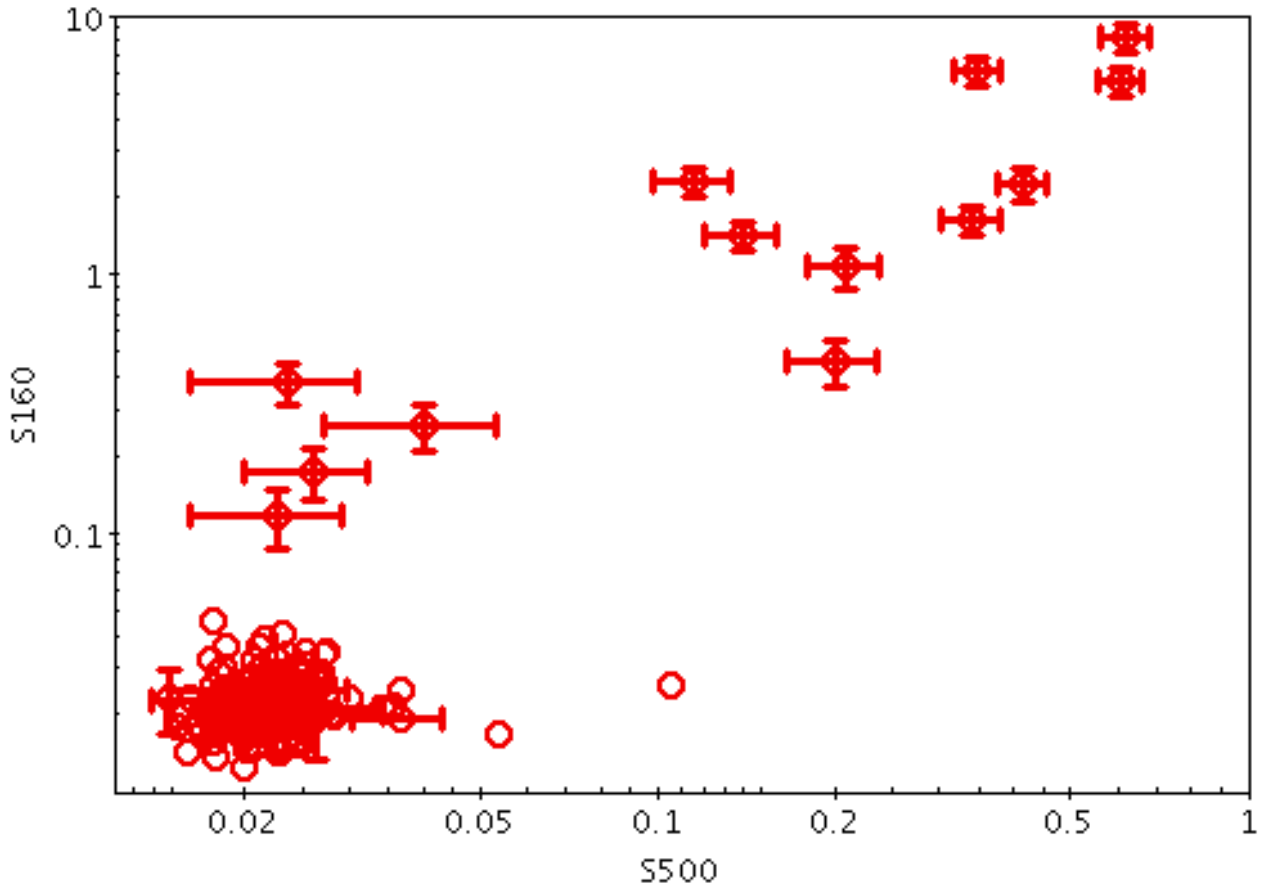
Plots symmetric or asymmetric error bars in some or all of the plot dimensions. The shape of the error "bars" is quite configurable, including (for 2-d and 3-d errors) ellipses, rectangles etc aligned with the axes.

Usage Overview:

```
layerN=xyerror errorbarN=none|lines|capped_lines|... thickN=<int-value>
               shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-param
               xN=<num-expr> yN=<num-expr> xerrhiN=<num-expr>
               xerrloN=<num-expr> yerrhiN=<num-expr> yerrloN=<num-expr>
               inN=<table> ifmtN=<in-format> istreamN=true|false
               icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in=J_MNRAS_440_1571.vot x=S500 y=S160
                  layer1=mark size1=5 shape1=fat_circle
                  layer2=xyerror xerrhi2=e_S500 yerrhi2=e_S160 errorbar2=capped_lines thick
                  xlog=true ylog=true shading=flat xmin=0.012 xmax=1 ymin=0.01 ymax=10
```

`errorbarN = none|lines|capped_lines|...` ***(MultiPointShape)***

How errorbars are represented.

The available options are:

- `none`
- `lines`
- `capped_lines`
- `caps`

- arrows
- ellipse
- crosshair_ellipse
- rectangle
- crosshair_rectangle
- filled_ellipse
- filled_rectangle

[Default: lines]

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter inN. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter inN. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value (auto) (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: (auto)]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "-", meaning standard input. In this case the input format must be given explicitly using the ifmtN parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form :<scheme-name>:<scheme-args>.
- A system command line with either a "<" character at the start, or a "|" character at the end ("<syscmd" or "syscmd|"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the inN parameter will be read as a stream. It is necessary to give the ifmtN parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: false]

shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>

(ShapeMode)
Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`thickN = <int-value>` ***(Integer)***
Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

`xN = <num-expr>` ***(String)***
Horizontal coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xerrhiN = <num-expr>` ***(String)***
Error in the X coordinate in the positive direction. If no corresponding negative error value is supplied, then this value is also used in the negative direction, i.e. in that case errors are assumed to be symmetric.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xerrloN = <num-expr>` ***(String)***
Error in the X coordinate in the negative direction. If left blank, it is assumed to take the same value as the positive error.

The value is a numeric algebraic expression based on column names as described in Section 10.

`yN = <num-expr>` ***(String)***
Vertical coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`yerrhiN = <num-expr>` ***(String)***
Error in the Y coordinate in the positive direction. If no corresponding negative error value is supplied, then this value is also used in the negative direction, i.e. in that case errors are assumed to be symmetric.

The value is a numeric algebraic expression based on column names as described in Section 10.

`yerrloN = <num-expr>` ***(String)***
Error in the Y coordinate in the negative direction. If left blank, it is assumed to take the same value as the positive error.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.6 `xyellipse`

Plots an ellipse (or rectangle, triangle, or other similar figure) defined by two principal radii and an optional angle of rotation, the so-called position angle. This angle, if specified, is in degrees and gives the angle counterclockwise from the horizontal axis to the first principal radius.

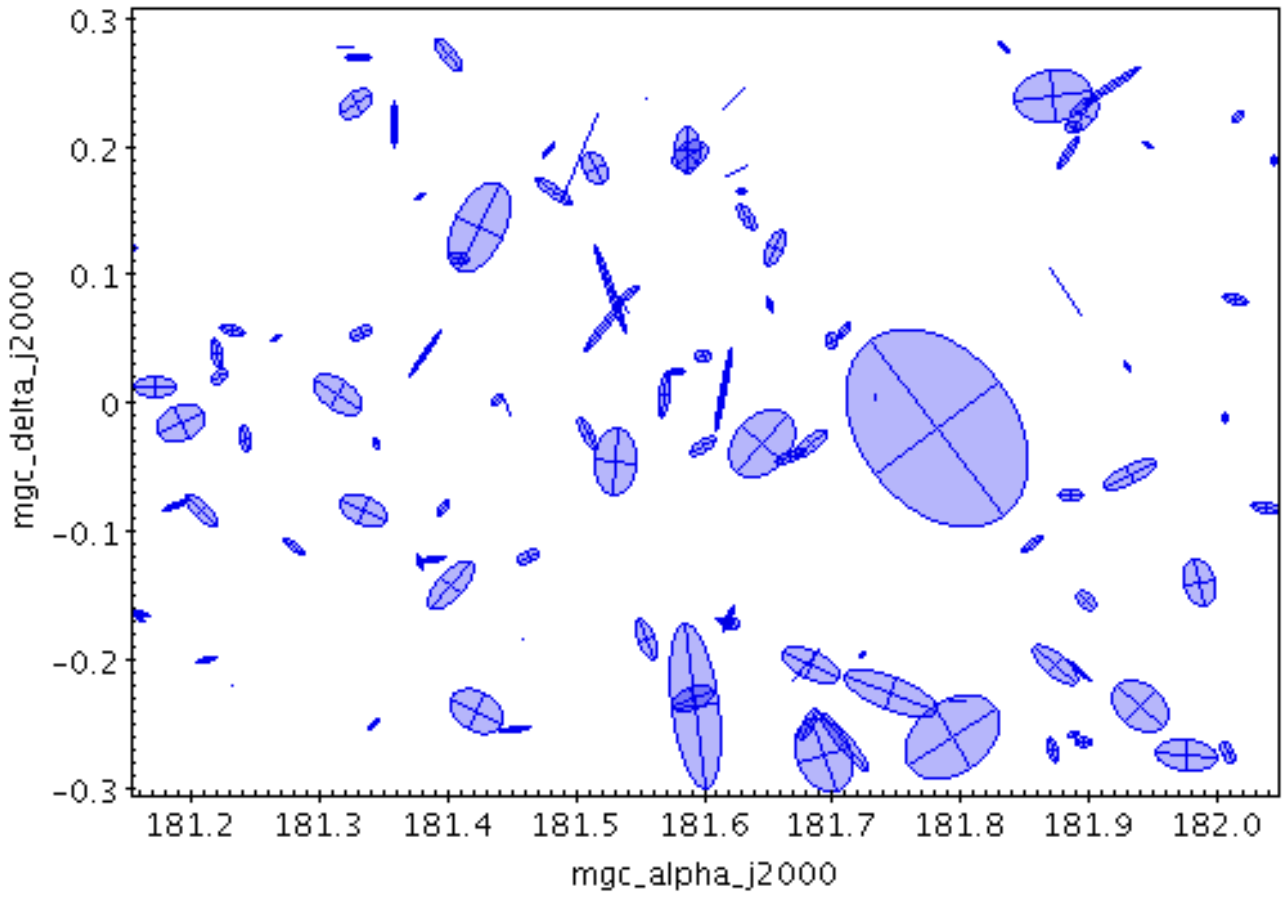
In some cases the supplied data values give the actual extents in data coordinates for the plotted ellipses but sometimes the data is on a different scale or in different units to the positional coordinates. As a convenience for this case, the plotter can optionally scale the magnitudes of all the ellipses to make them a reasonable size on the plot, so by default the largest ones are a few tens of pixels long. This auto-scaling is turned off by default, but it can be activated with the `autoscale` option. Whether autoscaling is on or off, the `scale` option can be used to apply a fixed scaling factor.

Usage Overview:

```
layerN=xyellipse ellipseN=ellipse|crosshair_ellipse|... thickN=<int-value>
                 scaleN=<factor> autoscaleN=true|false
                 shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-par
                 xN=<num-expr> yN=<num-expr> raN=<num-expr> rbN=<num-expr>
                 posangN=<deg-expr> inN=<table> ifmtN=<in-format>
                 istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:



```
stilts plot2plane in=mgc_ok.fits x=mgc_alpha_j2000 y=mgc_delta_j2000
                  ra=bulge_re/3600. rb=bulge_re*bulge_e/3600. posang=bulge_pa
                  autoscale=false scale=10 color=blue
                  layer1=xyellipse ellipse1=filled_ellipse shading1=transparent opaque1=4
                  layer2=xyellipse ellipse2=crosshair_ellipse
                  aspect=1 xmin=181.3 xmax=181.9
```

`autoscaleN = true|false` ***(Boolean)***

Determines whether the default size of variable-sized markers like vectors and ellipses are automatically scaled to have a sensible size. If true, then the sizes of all the plotted markers are examined, and some dynamically calculated factor is applied to them all to make them a sensible size (by default, the largest ones will be a few tens of pixels). If false, the sizes will be the actual input values interpreted in data coordinates.

If auto-scaling is on, then markers will keep approximately the same screen size during zoom operations; if it's off, they will keep the same size in data coordinates.

Marker size is also affected by the `scale` parameter.

[Default: `false`]

`ellipseN = ellipse|crosshair_ellipse|...` ***(MultiPointShape)***

How ellipses are represented.

The available options are:

- `ellipse`
- `crosshair_ellipse`
- `filled_ellipse`
- `rectangle`
- `crosshair_rectangle`
- `filled_rectangle`
- `open_triangle`
- `filled_triangle`
- `lines`
- `capped_lines`
- `arrows`

[Default: `ellipse`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`posangN = <deg-expr>` ***(String)***
Orientation of the ellipse. The value is the angle in degrees from the X axis towards the Y axis of the first principal axis of the ellipse.

The value is a numeric algebraic expression based on column names as described in Section 10.

`raN = <num-expr>` ***(String)***
Ellipse first principal radius.

The value is a numeric algebraic expression based on column names as described in Section 10.

`rbN = <num-expr>` ***(String)***
Ellipse second principal radius. If this value is blank, the two radii will be assumed equal, i.e. the ellipses will be circles.

The value is a numeric algebraic expression based on column names as described in Section 10.

`scaleN = <factor>` ***(Double)***
Affects the size of variable-sized markers like vectors and ellipses. The default value is 1, smaller or larger values multiply the visible sizes accordingly.

[Default: `1`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***
Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`

- translucent (Section 8.4.3)
- transparent (Section 8.4.4)
- density (Section 8.4.5)
- aux (Section 8.4.6)
- weighted (Section 8.4.7)

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: auto]

thickN = <int-value> ***(Integer)***
Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: 0]

xN = <num-expr> ***(String)***
Horizontal coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

yN = <num-expr> ***(String)***
Vertical coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.7 xycorr

Plots an error ellipse (or rectangle or other similar figure) defined by errors in the X and Y directions, and a correlation between the two errors.

The supplied correlation is a dimensionless value in the range -1..+1 and is equal to the covariance divided by the product of the X and Y errors. The covariance matrix is thus:

```
[  xerr*xerr         xerr*yerr*xycorr  ]
[  xerr*yerr*xycorr  yerr*yerr         ]
```

In some cases the supplied data values give the actual extents in data coordinates for the plotted ellipses but sometimes the data is on a different scale or in different units to the positional coordinates. As a convenience for this case, the plotter can optionally scale the magnitudes of all the ellipses to make them a reasonable size on the plot, so by default the largest ones are a few tens of pixels long. This auto-scaling is turned off by default, but it can be activated with the autoscale option. Whether autoscaling is on or off, the scale option can be used to apply a fixed scaling factor.

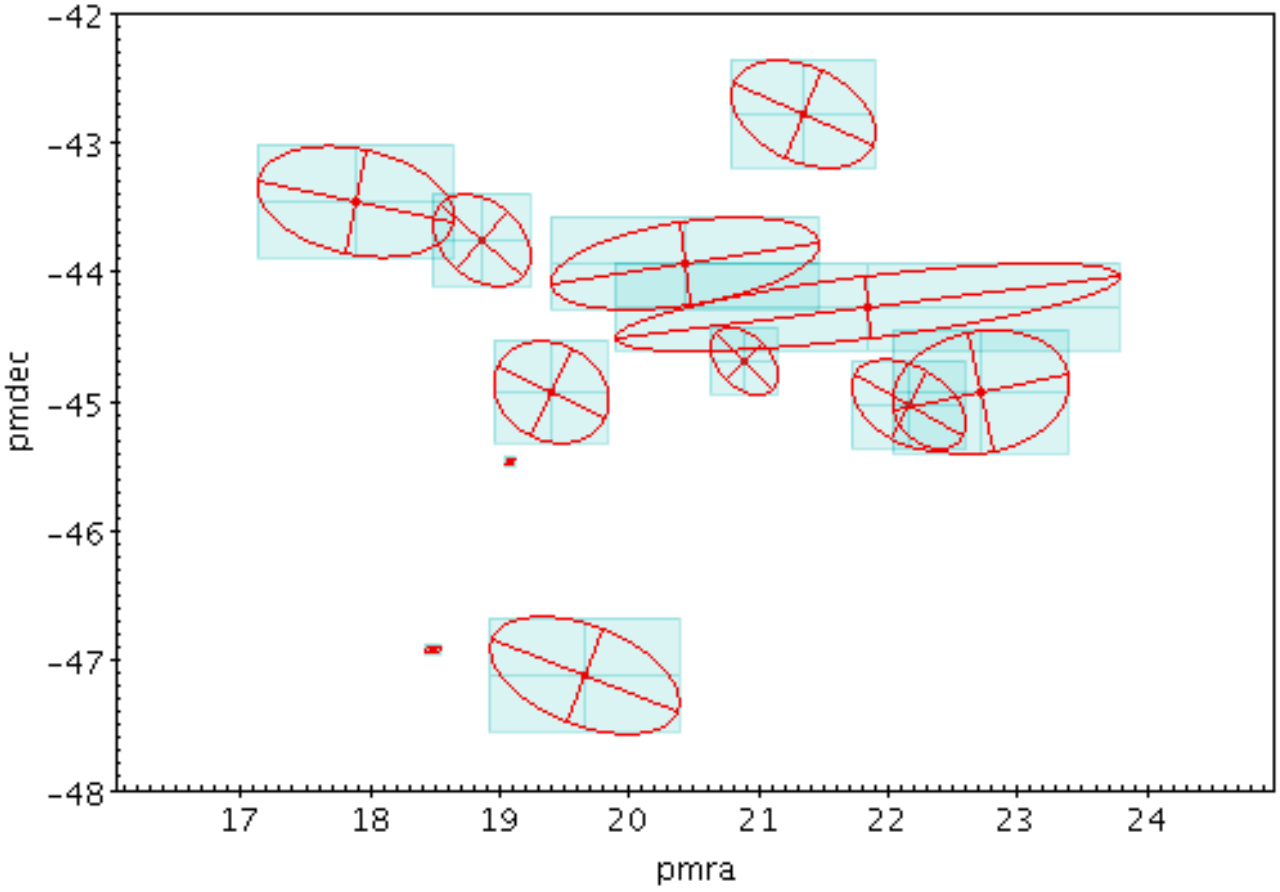
This plot type is suitable for use with the <x>_error and <x>_<y>_corr columns in the *Gaia* source catalogue.

Usage Overview:

```
layerN=xycorr ellipseN=ellipse|crosshair_ellipse|... thickN=<int-value>
              scaleN=<factor> autoscaleN=true|false
              shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-params
              xN=<num-expr> yN=<num-expr> xerrN=<num-expr> yerrN=<num-expr>
              xycorrN=<num-expr> inN=<table> ifmtN=<in-format>
              istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in=tgas_source.fits icmd='select skyDistanceDegrees(ra,dec,56.9,23.9)<0.4
                  x=pmra y=pmdec
                  layer1=mark
                  xerrhi2=pmra_error yerrhi2=pmdec_error
                  color2=cyan shading2=transparent
                  layer2a=xyerror errorbar2a=filled_rectangle opaque2a=10
                  layer2b=xyerror errorbar2b=crosshair_rectangle opaque2b=4
                  layer3=xycorr autoscale3=false
                  xerr3=pmra_error yerr3=pmdec_error xycorr3=pmra_pmdec_corr
                  ellipse3=crosshair_ellipse
                  aspect=1
                  xmin=17 xmax=24 ymin=-48 ymax=-42
```

`autoscaleN = true|false` ***(Boolean)***

Determines whether the default size of variable-sized markers like vectors and ellipses are automatically scaled to have a sensible size. If true, then the sizes of all the plotted markers are examined, and some dynamically calculated factor is applied to them all to make them a sensible size (by default, the largest ones will be a few tens of pixels). If false, the sizes will be the actual input values interpreted in data coordinates.

If auto-scaling is on, then markers will keep approximately the same screen size during zoom operations; if it's off, they will keep the same size in data coordinates.

Marker size is also affected by the `scale` parameter.

[Default: `false`]

`ellipseN = ellipse|crosshair_ellipse|...` ***(MultiPointShape)***

How ellipses are represented.

The available options are:

- `ellipse`
- `crosshair_ellipse`
- `filled_ellipse`
- `rectangle`
- `crosshair_rectangle`
- `filled_rectangle`
- `open_triangle`
- `filled_triangle`
- `lines`
- `capped_lines`
- `arrows`

[Default: `ellipse`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table

more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`scaleN = <factor>` ***(Double)***
Affects the size of variable-sized markers like vectors and ellipses. The default value is 1, smaller or larger values multiply the visible sizes accordingly.

[Default: `1`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***
Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`thickN = <int-value>` ***(Integer)***
Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

`xN = <num-expr>` ***(String)***
Horizontal coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xerrN = <num-expr>` ***(String)***
Error in the X coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xycorrN = <num-expr>` ***(String)***
Correlation beteween the errors in the X and Y directions. This is a dimensionless quantity in the range -1..+1, and is equivalent to the covariance divided by the product of the X and Y error values themselves. It corresponds to the `*_corr` values supplied in the Gaia source catalogue.

The value is a numeric algebraic expression based on column names as described in Section 10.

`yN = <num-expr>` ***(String)***
Vertical coordinate.

The value is a numeric algebraic expression based on column names as described in Section

10.

`yerrN = <num-expr>` ***(String)***
Error in the Y coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.8 `link2`

Plots a line linking two positions from the same input table row.

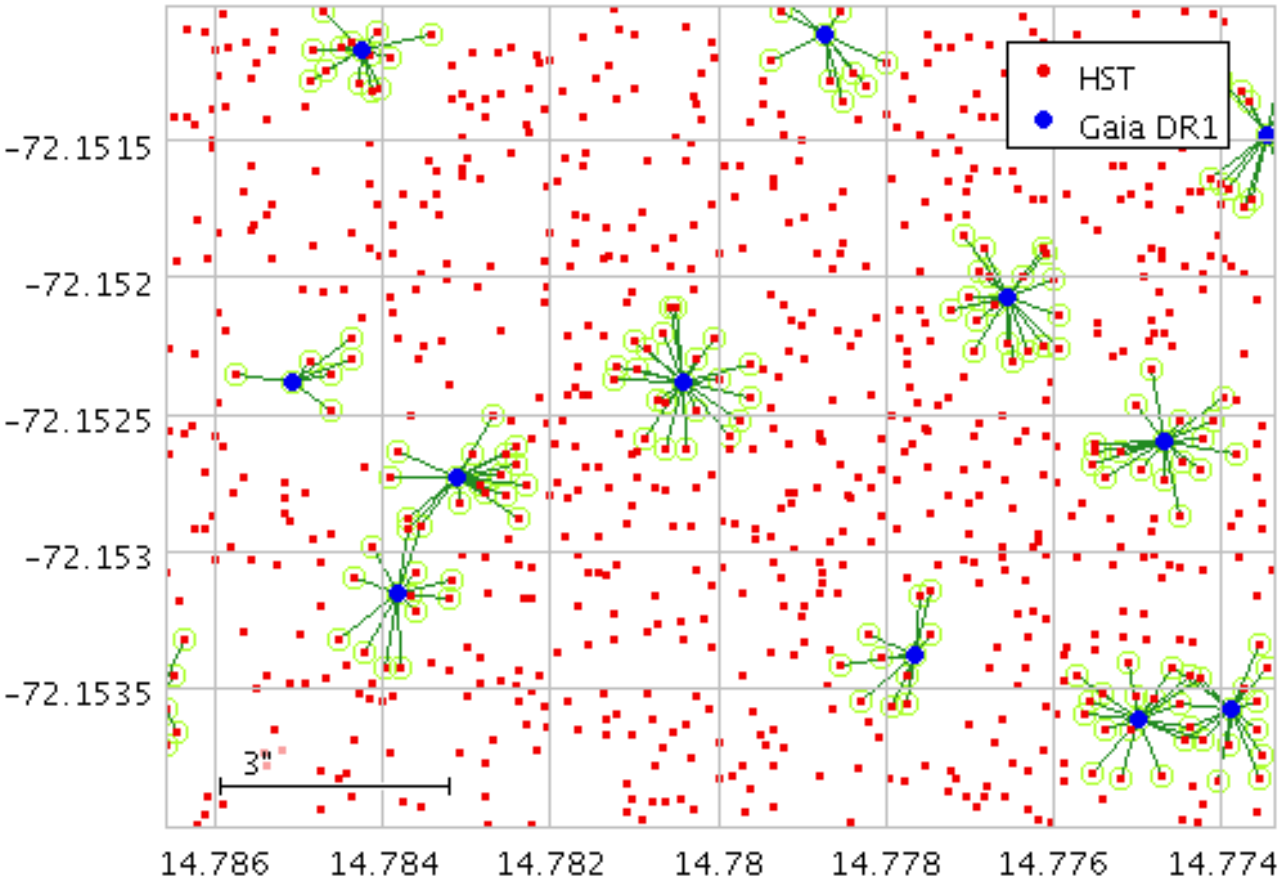
Usage Overview:

```
layerN=link2 thickN=<int-value>
             shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>
             <pos-coord-params1N> <pos-coord-params2N> inN=<table>
             ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:
The positional coordinates `<pos-coord-params1N>` , `<pos-coord-params2N>` give 2 positions for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `x1N`, `y1N`, `x2N` and `y2N`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2sky clon=14.78 clat=-72.1525 radius=0.0015 sex=false
               layer_h=mark in_h=ngc346.fits lon_h=_RAJ2000 lat_h=_DEJ2000 color_h=red
               layer_g=mark in_g=ngc346xGaiadr1.fits lon_g=ra lat_g=dec color_g=blue shadi
               in_x=ngc346xGaiadr1.fits lon1_x=_RAJ2000 lat1_x=_DEJ2000 lon2_x=ra lat2_x=d
               layer_xl=link2 color_xl=forestgreen
               layer_xm=mark2 color_xm=greenyellow size_xm=4 shape_xm=open_circle
               seq=_xm,_xl,_h,_g leglabel_h=HST leglabel_g='Gaia DR1' legseq=_h,_g legpos=
```

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`thickN = <int-value>` ***(Integer)***

Controls the line thickness used when drawing point-to-point links. Zero, the default value, means a 1-pixel-wide line is used, and larger values make drawn lines thicker.

[Default: `0`]

8.3.9 `mark2`

Plots 2 similar markers of fixed size and shape representing 2 separate positions from the same input table row. This is a convenience option that can be used with other plot layers based on 2 positions.

Usage Overview:

```
layerN=mark2 shapeN=filled_circle|open_circle|... sizeN=<pixels>
```

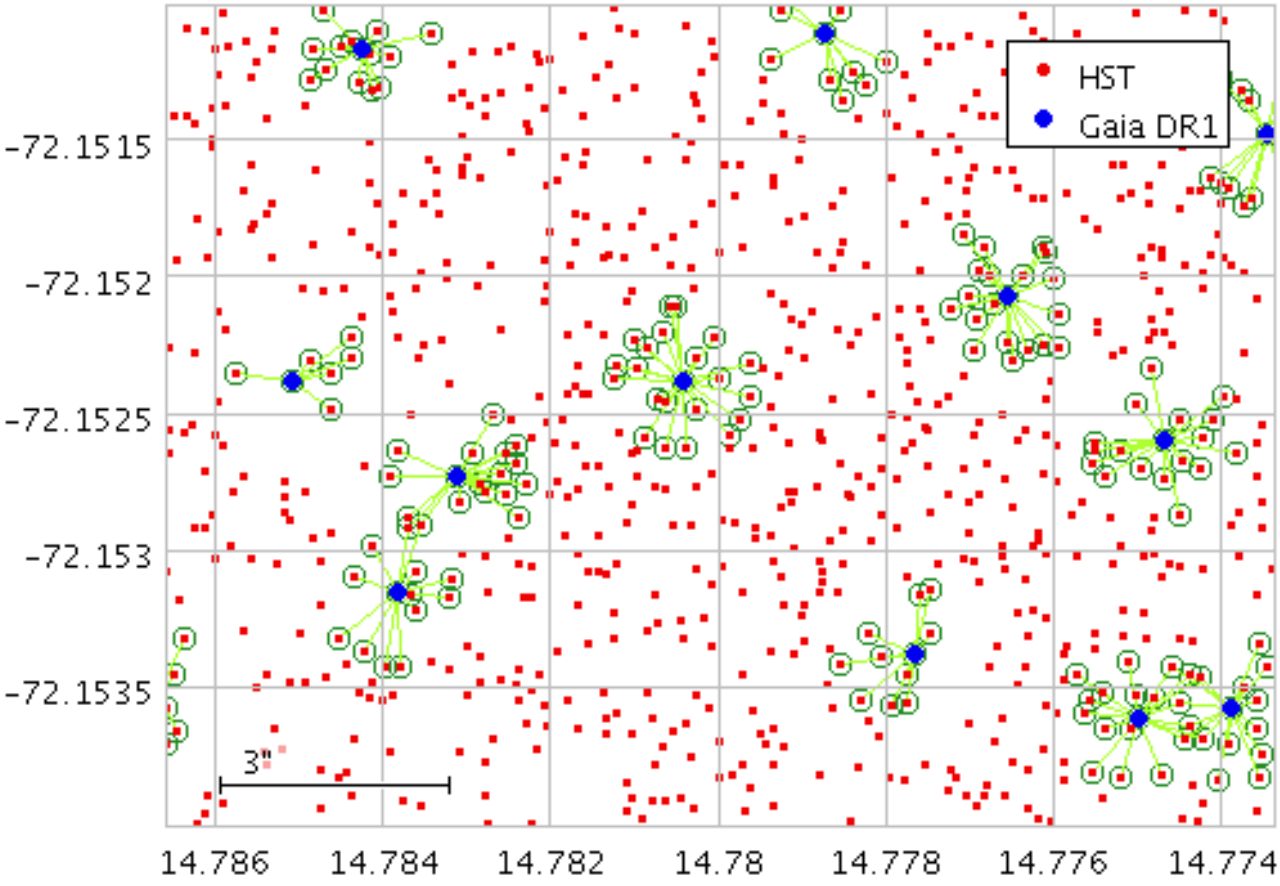
```
shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>
<pos-coord-params1N> <pos-coord-params2N> inN=<table>
ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:

The positional coordinates `<pos-coord-params1N>` , `<pos-coord-params2N>` give 2 positions for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `x1N`, `y1N`, `x2N` and `y2N`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2sky clon=14.78 clat=-72.1525 radius=0.0015 sex=false
               layer_h=mark in_h=ngc346.fits lon_h=_RAJ2000 lat_h=_DEJ2000 color_h=red
               layer_g=mark in_g=ngc346xGaiadr1.fits lon_g=ra lat_g=dec color_g=blue shadi
               in_x=ngc346xGaiadr1.fits lon1_x=_RAJ2000 lat1_x=_DEJ2000 lon2_x=ra lat2_x=d
               layer_xl=link2 color_xl=greenyellow
               layer_xm=mark2 color_xm=forestgreen size_xm=4 shape_xm=open_circle
               seq=_xm,_xl,_h,_g leglabel_h=HST leglabel_g='Gaia DR1' legseq=_h,_g legpos=
```

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps.

The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`shapeN = filled_circle|open_circle|...` ***(MarkerShape)***
Sets the shape of markers that are plotted at each position of the scatter plot.

The available options are:

- `filled_circle`
- `open_circle`
- `cross`
- `x`
- `open_square`
- `open_diamond`
- `open_triangle_up`
- `open_triangle_down`
- `fat_circle`
- `fat_cross`
- `fat_x`
- `fat_square`
- `fat_diamond`
- `fat_triangle_up`
- `fat_triangle_up`
- `filled_square`
- `filled_diamond`
- `filled_triangle_up`
- `filled_triangle_down`

[Default: `filled_circle`]

`sizeN = <pixels>` ***(Integer)***
Size of the scatter plot markers. The unit is pixels, in most cases the marker is approximately twice the size of the supplied value.

[Default: `1`]

8.3.10 `poly4`

Draws a closed quadrilateral given the coordinates of its vertices supplied as 4 separate positions. The way that the polygon is drawn (outline, fill etc) is determined using the `polymode` option.

Polygons smaller than a configurable threshold size in pixels are by default represented by a replacement marker, so the position of even a very small polygon is still visible on the screen.

Usage Overview:

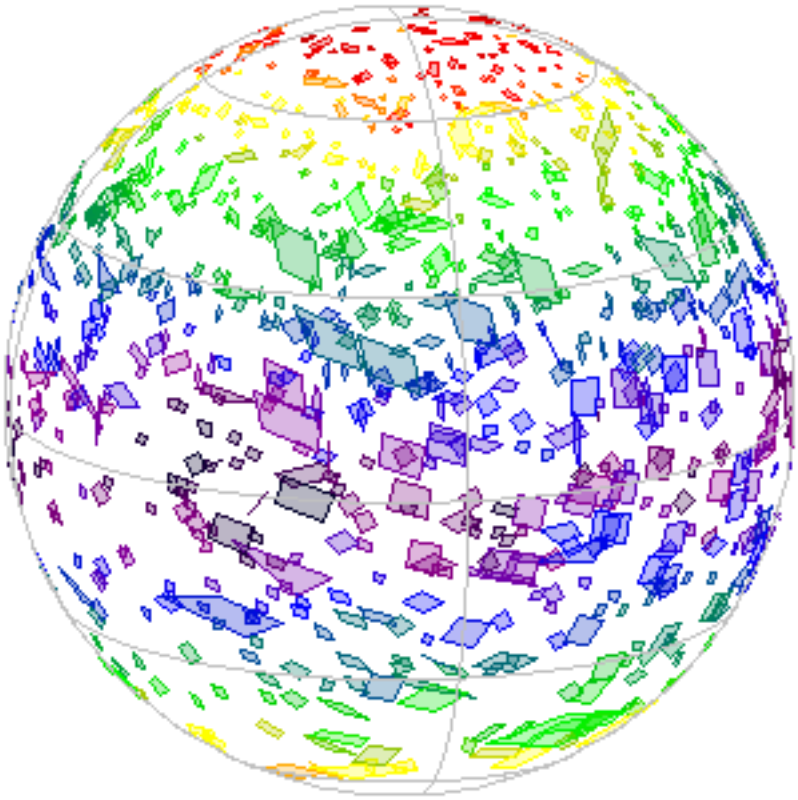
```
layerN=poly4 polymodeN=outline|border|fill|cross|star thickN=<int-value>
             minsizeN=<pixels> minshapeN=filled_circle|open_circle|...
             shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>
             <pos-coord-params1N> <pos-coord-params2N> <pos-coord-params3N>
             <pos-coord-params4N> inN=<table> ifmtN=<in-format>
             istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:
The positional coordinates `<pos-coord-params1N>` , `<pos-coord-params2N>` , `<pos-coord-params3N>` , `<pos-coord-params4N>` give 4 positions for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a

plane plot (`plot2plane`) the parameters would be `x1N`, `y1N`, `x2N`, `y2N`, `x3N`, `y3N`, `x4N` and `y4N`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:



```
stilts plot2sky in=big_tab_VIR_VIS_CSA_public.fits icmd='every 32'
               lon1=LON_CORNER_1 lat1=LAT_CORNER_1
               lon2=LON_CORNER_2 lat2=LAT_CORNER_2
               lon3=LON_CORNER_3 lat3=LAT_CORNER_3
               lon4=LON_CORNER_4 lat4=LAT_CORNER_4
               aux=RADIUS
               layer_o=poly4 polymode_o=outline shading_o=aux
               layer_f=poly4 polymode_f=fill shading_f=aux opaque_f=4
               auxmap=rainbow auxvisible=false xpix=300 ypix=300 labelpos=none
```

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the

table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`minshapeN = filled_circle|open_circle|...` ***(MarkerShape)***

Defines the shape of markers plotted instead of the actual polygon shape, for polygons that are smaller than the size threshold defined by `minsize`.

The available options are:

- `filled_circle`
- `open_circle`
- `cross`
- `x`
- `open_square`
- `open_diamond`
- `open_triangle_up`
- `open_triangle_down`
- `fat_circle`
- `fat_cross`
- `fat_x`
- `fat_square`
- `fat_diamond`
- `fat_triangle_up`
- `fat_triangle_up`
- `filled_square`
- `filled_diamond`
- `filled_triangle_up`
- `filled_triangle_down`

[Default: `x`]

`minsizeN = <pixels>` ***(Integer)***

Defines a threshold size in pixels below which, instead of the polygon defined by the other parameters, a replacement marker will be painted instead. If this is set to zero, then only the shape itself will be plotted, but if it is small it may appear as only a single pixel. By setting a larger value, you can ensure that the position of even small polygons is easily visible, at the expense of giving them an artificial shape and size. This value also defines the size of the replacement markers.

[Default: `1`]

`polymodeN = outline|border|fill|cross|star` ***(PolygonShape)***

Polygon drawing mode. Different options are available, including drawing an outline round the edge and filling the interior with colour.

The available options are:

- `outline`: draws a line round the outside of the polygon
- `border`: draws a line butting up to the outside of the polygon; may look better for adjacent shapes, but more expensive to draw
- `fill`: fills the interior of the polygon
- `cross`: draws a line round the outside of the polygon and lines between all the vertices
- `star`: draws a line round the outside of the polygon and lines from the nominal center to each vertex

[Default: `outline`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`thickN = <int-value>` ***(Integer)***

Controls the line thickness used when drawing polygons. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled polygons contain no line drawings.

[Default: `0`]

8.3.11 `mark4`

Plots 4 similar markers of fixed size and shape representing 4 separate positions from the same input table row. This is a convenience option that can be used with other plot layers based on 4 positions.

Usage Overview:

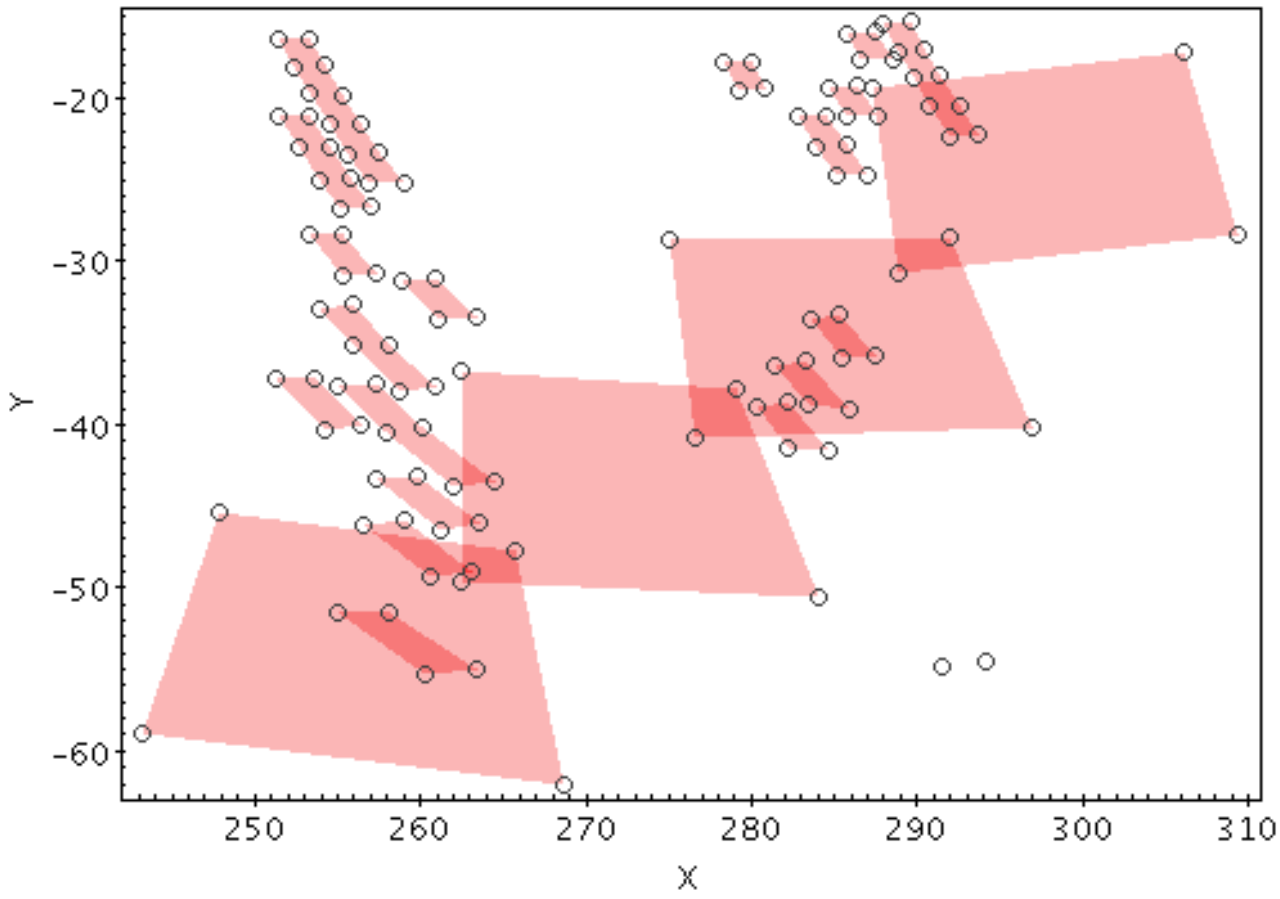
```
layerN=mark4 shapeN=filled_circle|open_circle|... sizeN=<pixels>
             shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN
             <pos-coord-params1N> <pos-coord-params2N> <pos-coord-params3N>
             <pos-coord-params4N> inN=<table> ifmtN=<in-format>
             istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:

The positional coordinates `<pos-coord-params1N>` , `<pos-coord-params2N>` , `<pos-coord-params3N>` , `<pos-coord-params4N>` give 4 positions for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `x1N`, `y1N`, `x2N`, `y2N`, `x3N`, `y3N`, `x4N` and `y4N`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2plane in=big_tab_VIR_VIS_CSA_public.fits
                  icmd='select IOF_055<0.005'
                  icmd='select lon_center>250&&lon_center<300&&lat_center>-65&&lat_center<-
                  x1=LON_CORNER_1 y1=LAT_CORNER_1
                  x2=LON_CORNER_2 y2=LAT_CORNER_2
                  x3=LON_CORNER_3 y3=LAT_CORNER_3
                  x4=LON_CORNER_4 y4=LAT_CORNER_4
                  layer_q=poly4 polymode_q=fill shading_q=transparent opaque_q=4
                  layer_m=mark4 color_m=404040 shape_m=open_circle size_m=3
```

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter inN. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter inN. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value (auto) (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: (auto)]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "-", meaning standard input. In this case the input format must be given explicitly using the ifmtN parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form :<scheme-name>:<scheme-args>.
- A system command line with either a "<" character at the start, or a "|" character at the end ("<syscmd" or "syscmd|"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the inN parameter will be read as a stream. It is necessary to give the ifmtN parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: false]

shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN> ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- auto (Section 8.4.1)
- flat (Section 8.4.2)
- translucent (Section 8.4.3)
- transparent (Section 8.4.4)
- density (Section 8.4.5)

- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`shapeN = filled_circle|open_circle|...` ***(MarkerShape)***
Sets the shape of markers that are plotted at each position of the scatter plot.

The available options are:

- `filled_circle`
- `open_circle`
- `cross`
- `x`
- `open_square`
- `open_diamond`
- `open_triangle_up`
- `open_triangle_down`
- `fat_circle`
- `fat_cross`
- `fat_x`
- `fat_square`
- `fat_diamond`
- `fat_triangle_up`
- `fat_triangle_up`
- `filled_square`
- `filled_diamond`
- `filled_triangle_up`
- `filled_triangle_down`

[Default: `filled_circle`]

`sizeN = <pixels>` ***(Integer)***
Size of the scatter plot markers. The unit is pixels, in most cases the marker is approximately twice the size of the supplied value.

[Default: `1`]

8.3.12 `polygon`

Draws a closed polygon given an array of coordinates that define its vertices. In fact this plot requires the position of the first vertex supplied as a positional value in the usual way (e.g. `X` and `Y` coordinates) and the second, third etc vertices supplied as an array using the `otherpoints` parameter.

Invocation might therefore look like "`xN=x1 yN=y1 otherpointsN=array(x2,y2, x3,y3, x4,y4)`".

Usage Overview:

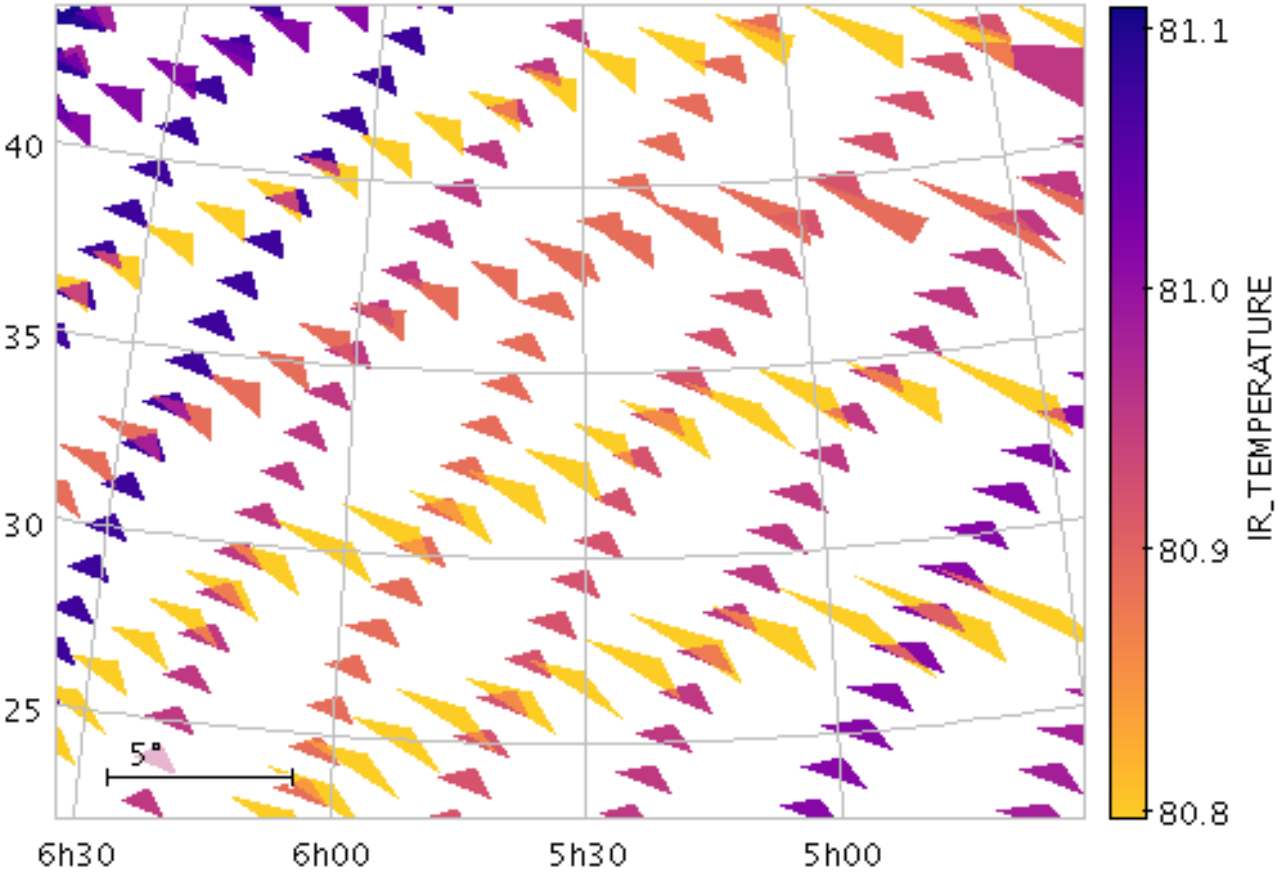
```
layerN=polygon useposN=true|false polymodeN=outline|border|fill|cross|star
               thickN=<int-value>
               shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-param
               <pos-coord-paramsN> otherpointsN=<array-expr> inN=<table>
               ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:

The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2sky in=big_tab_VIR_VIS_CSA_public.fits
               icmd='select ALTITUDE>4e4&&ALTITUDE<4.3e4'
               layer=polygon polymode=fill
               lon=LON_CENTER lat=LAT_CENTER
               otherpoints=array(lon_corner_1,lat_corner_1,lon_corner_2,lat_corner_2)
               shading=weighted weight=IR_TEMPERATURE auxmap=plasma
               clon=83 clat=34 radius=11
```

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter

commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`otherpointsN = <array-expr>` ***(String)***

Array of coordinates giving the points of the vertices defining the polygon to be drawn. These coordinates are given as an interleaved array by this parameter, e.g. (x1,y1, x2,y2, y3,y3). The basic position for the row being plotted either is or is not included as the first vertex, according to the setting of the `usepos` parameter.

Some expression language functions that can be useful when specifying this parameter are `array()` and `parseDoubles()`.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`polymodeN = outline|border|fill|cross|star` ***(PolygonShape)***

Polygon drawing mode. Different options are available, including drawing an outline round the edge and filling the interior with colour.

The available options are:

- `outline`: draws a line round the outside of the polygon
- `border`: draws a line butting up to the outside of the polygon; may look better for adjacent shapes, but more expensive to draw
- `fill`: fills the interior of the polygon

- `cross`: draws a line round the outside of the polygon and lines between all the vertices
- `star`: draws a line round the outside of the polygon and lines from the nominal center to each vertex

[Default: `outline`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`thickN = <int-value>` ***(Integer)***

Controls the line thickness used when drawing polygons. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled polygons contain no line drawings.

[Default: `0`]

`useposN = true|false` ***(Boolean)***

Determines whether the basic positional coordinates are included as one of the polygon vertices or not. The polygon has N+1 vertices if true, or N vertices if false, where N is the number of vertices supplied by the array coordinate. If false, the basic position is ignored for the purposes of drawing the polygon.

[Default: `true`]

8.3.13 `area`

Plots a region on the plotting surface specified by a string or array of numbers. The area may be specified as an STC-S string (as for example in an ObsCore or EPN-TAP `s_region` column) or using an array of numbers representing a polygon, circle or point as flagged using the DALI/VOTable extended type (xtype) marker, or as an ASCII-encoded MOC.

Areas smaller than a configurable threshold size in pixels are by default represented by a replacement marker, so the position of even a very small area is still visible on the screen.

This plot type is generally intended for displaying relatively small shapes such as instrument footprints. It can be used for areas that are larger as well, but there may be issues with use, for instance auto-determination of the initial plot region may not work so well, and the rendering of shapes that are large relative to the sky may be inaccurate. These issues may be addressed in future releases.

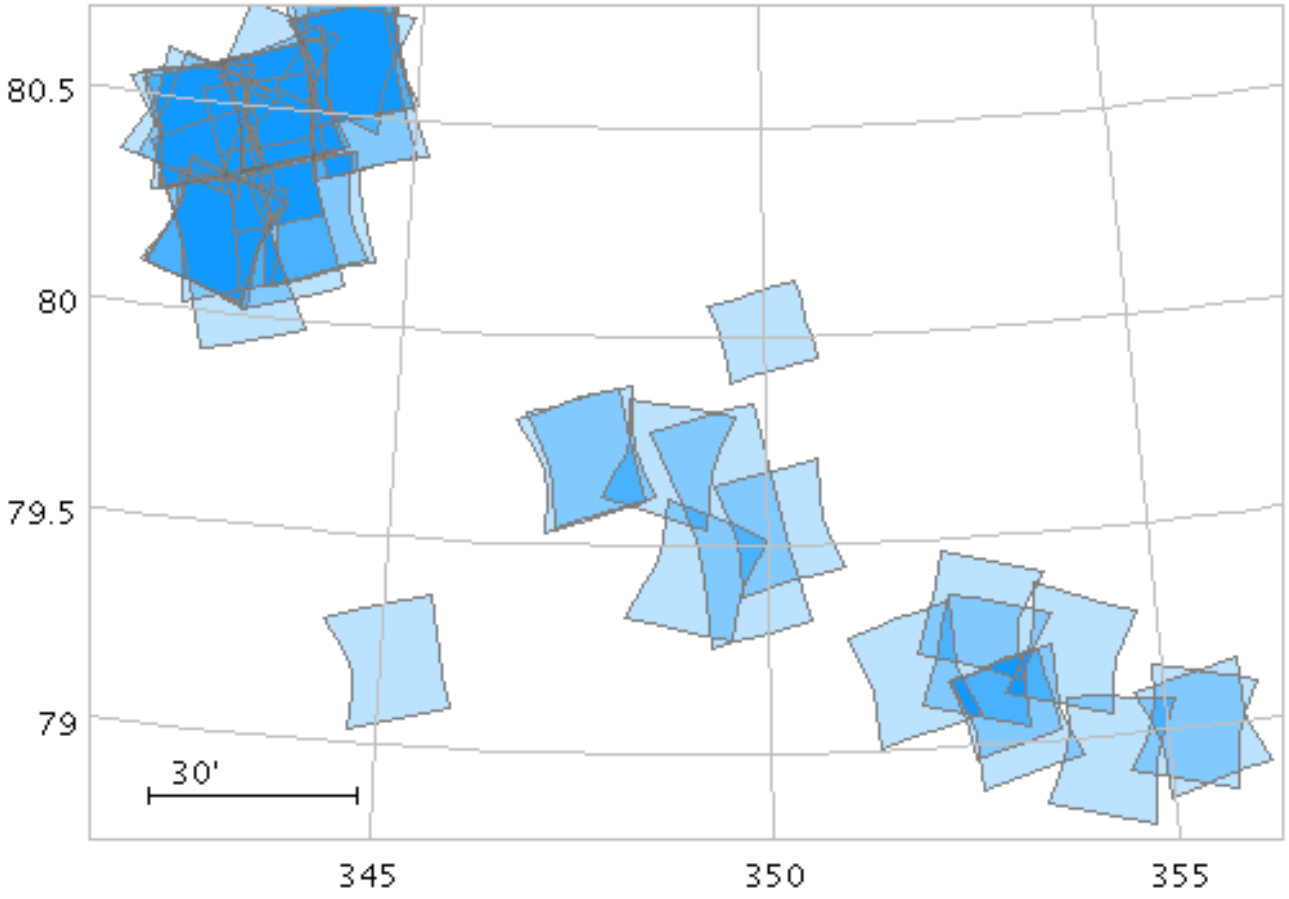
Usage Overview:

```
layerN=area polymodeN=outline|border|fill|cross|star thickN=<int-value>
```

```
minsizeN=<pixels> minshapeN=filled_circle|open_circle|...
shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>
areaN=<area-expr>
areatypeN=STC-S|POLYGON|CIRCLE|POINT|MOC-ASCII|UNIQ inN=<table>
ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2sky reflectlon=false sex=false clon=348.9 clat=79.8 radius=1.0
               in=crism.fits icmd='select sensor_id==0x4c'
               area_p=s_region areatype_p=stc-s
               layer_pf=area polymode_pf=fill color_pf=1199ff shading_pf=transparent
               layer_pl=area polymode_pl=outline color_pl=grey
```

`areaN = <area-expr>` ***(String)***

Expression giving the geometry of a 2D region on the plot. It may be a string- or array-valued expression, and its interpretation depends on the value of the corresponding `areatype` parameter.

The value is a Area value algebraic expression based on column names as described in Section 10.

`areatypeN = STC-S|POLYGON|CIRCLE|POINT|MOC-ASCII|UNIQ` ***(AreaMapper)***

Selects the form in which the Area value for parameter `areaN` is supplied. Options are:

- `STC-S`: Region description using STC-S syntax; see TAP 1.0, section 6. Note there are some restrictions: `<frame>`, `<refpos>` and `<flavor>` metadata are ignored, polygon winding direction is ignored (small polygons are assumed) and the `INTERSECTION` and `NOT` constructions are not supported. The non-standard `MOC` construction is supported.

- `POLYGON`: 2n-element array (`x1`,`y1`, `x2`,`y2`,...); a `NaN`,`NaN` pair can be used to delimit distinct polygons.
- `CIRCLE`: 3-element array (`x`,`y`,`r`)
- `POINT`: 2-element array (`x`,`y`)
- `MOC-ASCII`: Region description using ASCII MOC syntax; see MOC 1.1 2.3.2. Note there are currently a few issues with MOC plotting, especially for large pixels.
- `UNIQ`: Region description representing a single HEALPix cell as defined by an UNIQ value, see MOC 1.1 sec 2.3.1.

If left blank, a guess will be taken depending on the data type of the value supplied for the `areaN` value.

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`minshapeN = filled_circle|open_circle|...` ***(MarkerShape)***

Defines the shape of markers plotted instead of the actual polygon shape, for polygons that are smaller than the size threshold defined by `minsize`.

The available options are:

- `filled_circle`
- `open_circle`
- `cross`
- `x`
- `open_square`
- `open_diamond`
- `open_triangle_up`
- `open_triangle_down`
- `fat_circle`
- `fat_cross`
- `fat_x`
- `fat_square`
- `fat_diamond`
- `fat_triangle_up`
- `fat_triangle_up`
- `filled_square`
- `filled_diamond`
- `filled_triangle_up`
- `filled_triangle_down`

[Default: `x`]

`minsizeN = <pixels>` ***(Integer)***

Defines a threshold size in pixels below which, instead of the polygon defined by the other parameters, a replacement marker will be painted instead. If this is set to zero, then only the shape itself will be plotted, but if it is small it may appear as only a single pixel. By setting a larger value, you can ensure that the position of even small polygons is easily visible, at the expense of giving them an artificial shape and size. This value also defines the size of the replacement markers.

[Default: `1`]

`polymodeN = outline|border|fill|cross|star` ***(PolygonShape)***

Polygon drawing mode. Different options are available, including drawing an outline round the edge and filling the interior with colour.

The available options are:

- `outline`: draws a line round the outside of the polygon
- `border`: draws a line butting up to the outside of the polygon; may look better for adjacent shapes, but more expensive to draw
- `fill`: fills the interior of the polygon
- `cross`: draws a line round the outside of the polygon and lines between all the vertices
- `star`: draws a line round the outside of the polygon and lines from the nominal center to each vertex

[Default: `outline`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- auto (Section 8.4.1)
- flat (Section 8.4.2)
- translucent (Section 8.4.3)
- transparent (Section 8.4.4)
- density (Section 8.4.5)
- aux (Section 8.4.6)
- weighted (Section 8.4.7)

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: auto]

thickN = <int-value> ***(Integer)***

Controls the line thickness used when drawing polygons. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled polygons contain no line drawings.

[Default: 0]

8.3.14 central

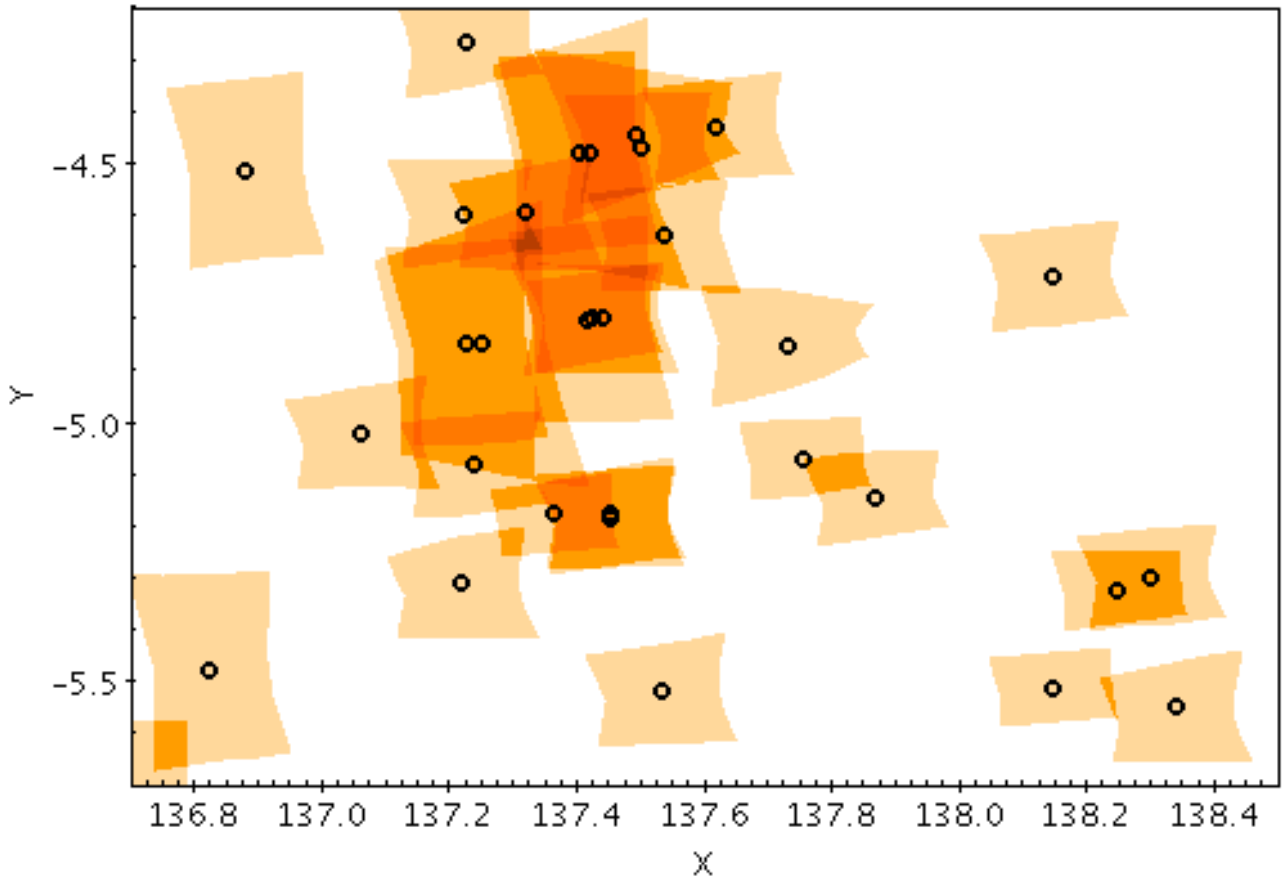
Plots the nominal central point of an area. This appears just like a normal marker plot, but can be used when the available geometry information is an area description (such as an STC-S string or an array of polygon vertices) rather than coordinate values such as an X,Y pair. The position plotted is the nominal center of the shape as determined by the plotting code; that may or may not correspond to the actual center.

Usage Overview:

```
layerN=central shapeN=filled_circle|open_circle|... sizeN=<pixels>
               shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-param
               areaN=<area-expr>
               areatypeN=STC-S|POLYGON|CIRCLE|POINT|MOC-ASCII|UNIQ
               inN=<table> ifmtN=<in-format> istreamN=true|false
               icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix N.

Example:




```
stilts plot2plane xmin=136.7 xmax=138.5 ymin=-5.7 ymax=-4.2
                  in=crism.fits icmd='select sensor_id==0x53'
                  area=s_region areatype=STC-S
                  layer1=area polymode1=fill shading1=density densemap1=heat
                  layer2=central shape2=fat_circle size2=3 color2=black
```

areaN = <area-expr> ***(String)***

Expression giving the geometry of a 2D region on the plot. It may be a string- or array-valued expression, and its interpretation depends on the value of the corresponding areatype parameter.

The value is a Area value algebraic expression based on column names as described in Section 10.

areatypeN = STC-S|POLYGON|CIRCLE|POINT|MOC-ASCII|UNIQ ***(AreaMapper)***

Selects the form in which the Area value for parameter areaN is supplied. Options are:

- STC-S: Region description using STC-S syntax; see TAP 1.0, section 6. Note there are some restrictions: <frame>, <refpos> and <flavor> metadata are ignored, polygon winding direction is ignored (small polygons are assumed) and the INTERSECTION and NOT constructions are not supported. The non-standard MOC construction is supported.
- POLYGON: 2n-element array (x1,y1, x2,y2,...); a NaN,NaN pair can be used to delimit distinct polygons.
- CIRCLE: 3-element array (x,y,r)
- POINT: 2-element array (x,y)
- MOC-ASCII: Region description using ASCII MOC syntax; see MOC 1.1 2.3.2. Note there are currently a few issues with MOC plotting, especially for large pixels.
- UNIQ: Region description representing a single HEALPix cell as defined by an UNIQ value, see MOC 1.1 sec 2.3.1.

If left blank, a guess will be taken depending on the data type of the value supplied for the

areaN value.

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter inN. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter inN. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value (auto) (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: (auto)]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "-", meaning standard input. In this case the input format must be given explicitly using the ifmtN parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form :<scheme-name>:<scheme-args>.
- A system command line with either a "<" character at the start, or a "|" character at the end ("<syscmd" or "syscmd|"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the inN parameter will be read as a stream. It is necessary to give the ifmtN parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: false]

shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN> ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- auto (Section 8.4.1)
- flat (Section 8.4.2)
- translucent (Section 8.4.3)

- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`shapeN = filled_circle|open_circle|...` ***(MarkerShape)***
Sets the shape of markers that are plotted at each position of the scatter plot.

The available options are:

- `filled_circle`
- `open_circle`
- `cross`
- `x`
- `open_square`
- `open_diamond`
- `open_triangle_up`
- `open_triangle_down`
- `fat_circle`
- `fat_cross`
- `fat_x`
- `fat_square`
- `fat_diamond`
- `fat_triangle_up`
- `fat_triangle_up`
- `filled_square`
- `filled_diamond`
- `filled_triangle_up`
- `filled_triangle_down`

[Default: `filled_circle`]

`sizeN = <pixels>` ***(Integer)***
Size of the scatter plot markers. The unit is pixels, in most cases the marker is approximately twice the size of the supplied value.

[Default: `1`]

8.3.15 `lines`

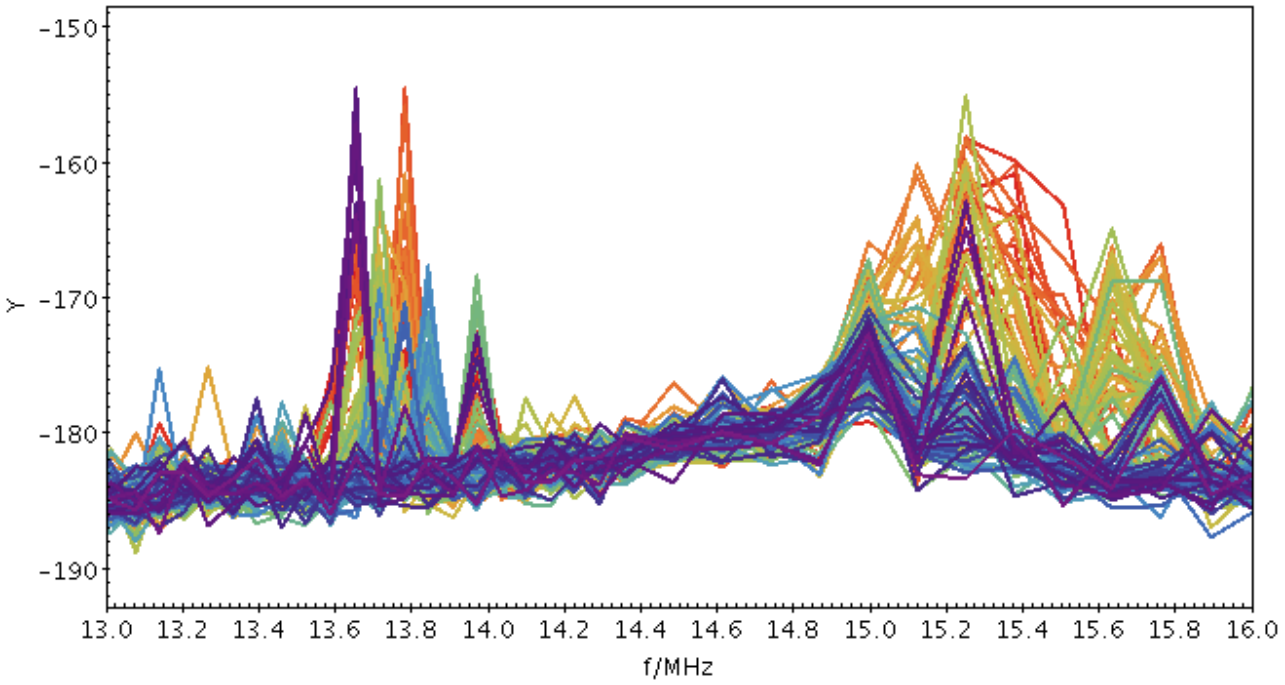
Plots an *N*-segment line for each input row, with the X and Y coordinate arrays each supplied by an *N*-element array value.

Usage Overview:

```
layerN=lines thickN=<pixels> dashN=dot|dash|...|<a,b,...>
             antialiasN=true|false
             shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN
             xsN=<array-expr> ysN=<array-expr> inN=<table> ifmtN=<in-format>
             istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in1=LRS_NPW_V010_20071101.cdf
                  layer1=lines xs=multiply(param$frequency,1e-6) xlabel=f/MHz ys=RX1 thick=
                  shading=aux aux=Epoch auxmap=sron
                  icmd='every 100' xmin=13 xmax=16 xpix=660 auxvisible=false
```

`antialiasN = true|false` ***(Boolean)***

If true, plotted lines are drawn with antialising. Antialised lines look smoother, but may take perceptibly longer to draw. Only has any effect for bitmapped output formats.

[Default: `false`]

`dashN = dot|dash|...|<a,b,...>` ***(float[])***

Determines the dash pattern of the line drawn. If null (the default), the line is solid.

Possible values for dashed lines are `dot`, `dash`, `longdash`, `dotdash`. You can alternatively supply a comma-separated list of on/off length values such as "`4,2,8,2`".

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "-", meaning standard input. In this case the input format must be given explicitly using the ifmtN parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form :<scheme-name>:<scheme-args>.
- A system command line with either a "<" character at the start, or a "|" character at the end ("<syscmd" or "syscmd|"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the inN parameter will be read as a stream. It is necessary to give the ifmtN parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: false]

shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN> ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- auto (Section 8.4.1)
- flat (Section 8.4.2)
- translucent (Section 8.4.3)
- transparent (Section 8.4.4)
- density (Section 8.4.5)
- aux (Section 8.4.6)
- weighted (Section 8.4.7)

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: auto]

thickN = <pixels> ***(Integer)***

Thickness of plotted line in pixels.

[Default: 1]

xsN = <array-expr> ***(String)***

Array giving the X coordinate array for each line. In most cases, if a blank value is supplied but Y values are present then a suitable linear sequence, of the same length as the Y array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

ysN = <array-expr> ***(String)***

Array giving the Y coordinate array for each line. In most cases, if a blank value is supplied but X values are present then a suitable linear sequence, of the same length as the X array, is

assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

8.3.16 `marks`

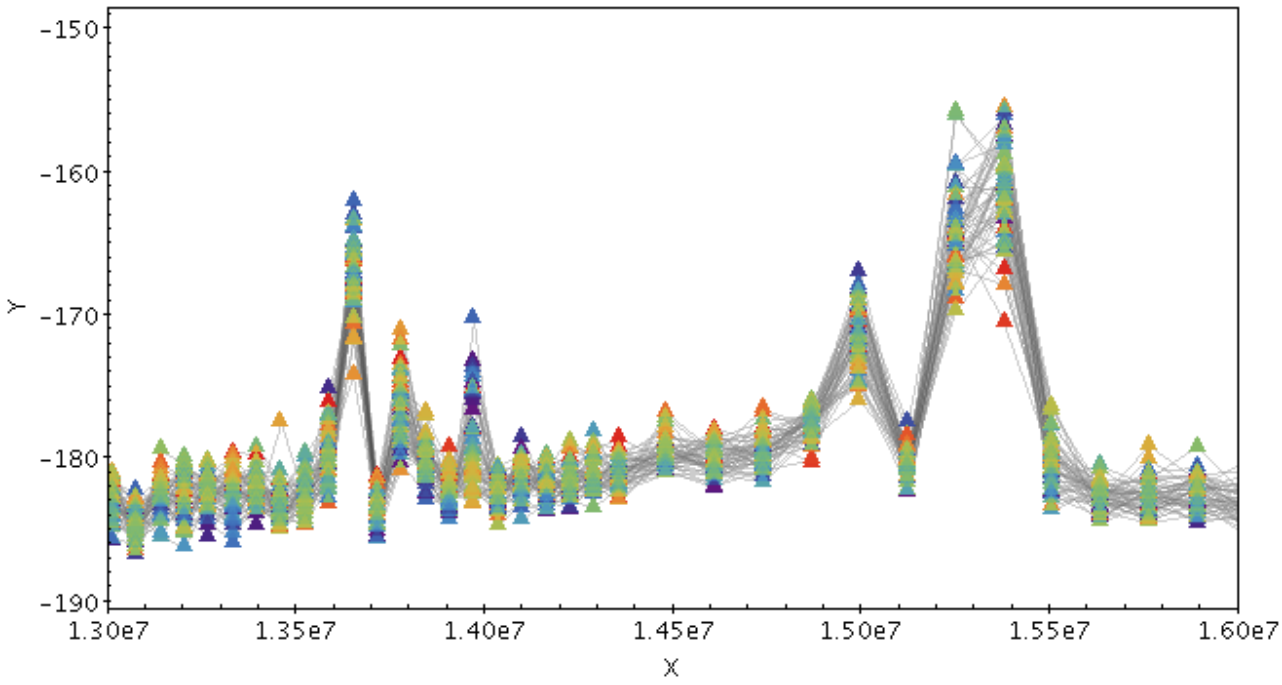
Plots *N* markers for each input row, with the X and Y coordinate values each supplied by an *N*-element array value.

Usage Overview:

```
layerN=marks shapeN=filled_circle|open_circle|... sizeN=<pixels>
             shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN
             xsN=<array-expr> ysN=<array-expr> inN=<table> ifmtN=<in-format>
             istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in=LRS_NPW_V010_20071101.cdf xs=param$frequency ys=RX2
                  layer1=lines shading1=density densemap1=greyscale denseclip1=0.2,0.7
                  layer2=marks shading2=weighted weight2=epoch shape2=filled_triangle_down
                  xmin=13e6 xmax=16e6 xpix=660 icmd='head 50' auxmap=sron auxvisible=false
```

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter

commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN> ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

shapeN = filled_circle|open_circle|... ***(MarkerShape)***

Sets the shape of markers that are plotted at each position of the scatter plot.

The available options are:

- `filled_circle`
- `open_circle`
- `cross`
- `x`
- `open_square`
- `open_diamond`
- `open_triangle_up`
- `open_triangle_down`
- `fat_circle`
- `fat_cross`
- `fat_x`
- `fat_square`
- `fat_diamond`
- `fat_triangle_up`
- `fat_triangle_up`
- `filled_square`
- `filled_diamond`
- `filled_triangle_up`
- `filled_triangle_down`

[Default: `filled_circle`]

`sizeN = <pixels>` ***(Integer)***
Size of the scatter plot markers. The unit is pixels, in most cases the marker is approximately twice the size of the supplied value.

[Default: `1`]

`xsN = <array-expr>` ***(String)***
Array giving the X coordinate array for each line. In most cases, if a blank value is supplied but Y values are present then a suitable linear sequence, of the same length as the Y array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`ysN = <array-expr>` ***(String)***
Array giving the Y coordinate array for each line. In most cases, if a blank value is supplied but X values are present then a suitable linear sequence, of the same length as the X array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

8.3.17 `yerrors`

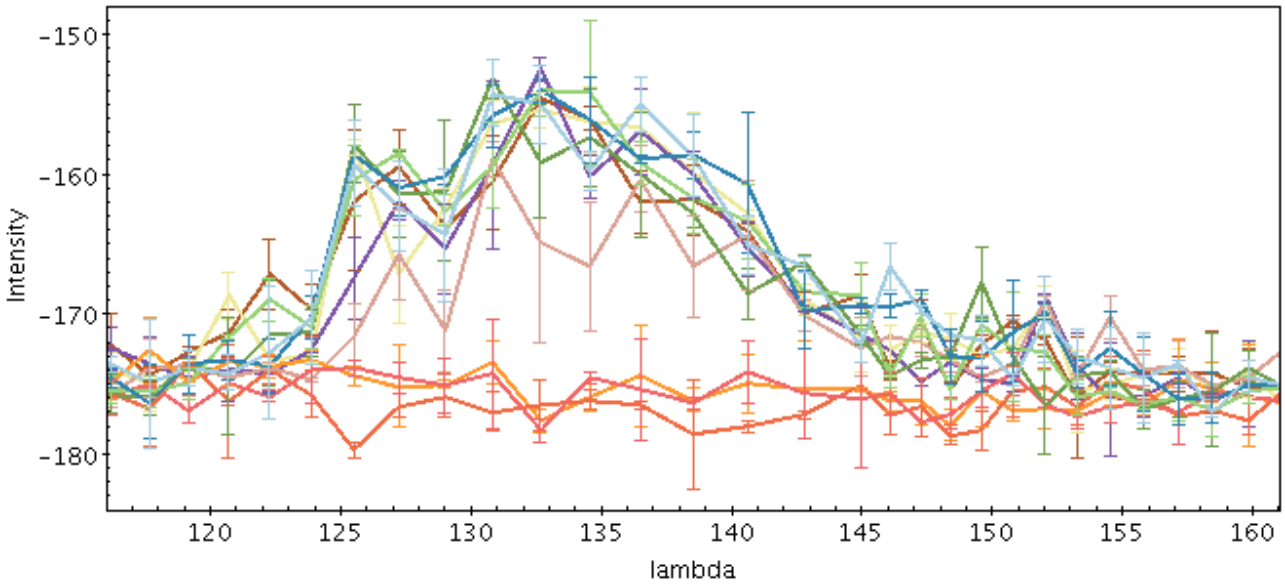
Plots *N* error bars in the Y direction for each input row, with the X, Y and error bar extents each supplied by *N*-element array values.

Usage Overview:

```
layerN=yerrors errorbarN=none|lines|capped_lines|caps|arrows
               thickN=<int-value>
               shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-param
               xsN=<array-expr> ysN=<array-expr> yerrhisN=<array-expr>
               yerrlosN=<array-expr> inN=<table> ifmtN=<in-format>
               istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in=LRS_NPW_V010_20071101.cdf
                  shading=aux aux=epoch
                  xs=divide(2.998e8,param$Frequency) ys=multiply(add(RX1,RX2),0.5)
                  layer_l=lines thick_l=2
                  layer_e=yerrors yerrhis_e=arrayFunc("abs(x)",subtract(RX1,RX2)) errorbar_
                  auxmap=paired auxvisible=false
                  xmin=116 xmax=161 ymin=-184 ymax=-148 xpix=660 ypix=300 icmd='every 1000'
                  xlabel=lambda ylabel=Intensity
```

`errorbarN = none|lines|capped_lines|caps|arrows` ***(MultiPointShape)***

How errorbars are represented.

The available options are:

- `none`
- `lines`
- `capped_lines`
- `caps`
- `arrows`

[Default: `lines`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are

listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value `"-"`, meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a `"<"` character at the start, or a `"|"` character at the end (`"<syscmd"` or `"syscmd|"`). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`thickN = <int-value>` ***(Integer)***

Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

`xsN = <array-expr>` ***(String)***

Array giving the X coordinate array for each line. In most cases, if a blank value is supplied but Y values are present then a suitable linear sequence, of the same length as the Y array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`yerrhisN = <array-expr>` ***(String)***

Array of errors in the Y coordinates in the positive direction. If no corresponding negative value is supplied, then this value is also used in the negative direction, i.e. in that case errors are assumed to be symmetric. Error exents must be positive; negative array elements are ignored.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`yerrlosN = <array-expr>` ***(String)***

Array of errors in the Y coordinates in the negative direction. If left blank, it is assumed to take the same value as in the positive direction. Error extents must be positive; negative array elements are ignored.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`ysN = <array-expr>` ***(String)***

Array giving the Y coordinate array for each line. In most cases, if a blank value is supplied but X values are present then a suitable linear sequence, of the same length as the X array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

8.3.18 `xyerrors`

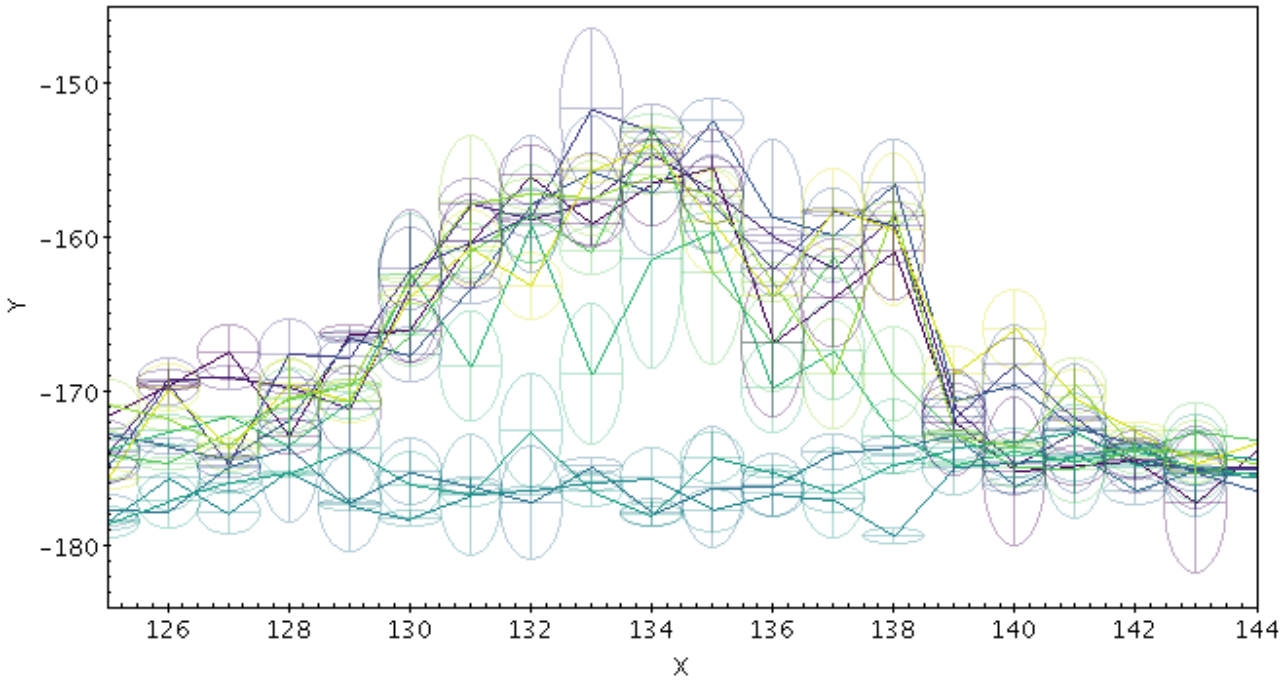
Plots *N* error bars in the X and Y directions for each input row, with the X, Y and error bar extents each supplied by *N*-element array values.

Usage Overview:

```
layerN=xyerrors errorbarN=none|lines|capped_lines|... thickN=<int-value>
                shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-para
                xsN=<array-expr> ysN=<array-expr> xerrhisN=<array-expr>
                xerrlosN=<array-expr> yerrhisN=<array-expr>
                yerrlosN=<array-expr> inN=<table> ifmtN=<in-format>
                istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in=LRS_NPW_V010_20071101.cdf ys=RX1
                  shading=aux aux=epoch
                  layer_1=lines opaque_1=1
                  layer_xy=xyerrors opaque_xy=3.3 errorbar_xy=crosshair_ellipse
                  xerrhis_xy=constant(512,0.5) yerrhis_xy=arrayFunc("abs(x)",subtract(RX1,R
                  xmin=125 xmax=144 ymin=-184 ymax=-145 xpix=660 icmd='every 1000'
                  auxmap=viridis auxvisible=false
```

`errorbarN = none|lines|capped_lines|...` ***(MultiPointShape)***

How errorbars are represented.

The available options are:

- `none`
- `lines`
- `capped_lines`
- `caps`
- `arrows`
- `ellipse`
- `crosshair_ellipse`
- `rectangle`
- `crosshair_rectangle`
- `filled_ellipse`
- `filled_rectangle`

[Default: `lines`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters

and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN> ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

thickN = <int-value> ***(Integer)***

Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

`xerrhisN = <array-expr>` ***(String)***
Array of errors in the X coordinates in the positive direction. If no corresponding negative value is supplied, then this value is also used in the negative direction, i.e. in that case errors are assumed to be symmetric. Error exents must be positive; negative array elements are ignored.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`xerrlosN = <array-expr>` ***(String)***
Array of errors in the X coordinates in the negative direction. If left blank, it is assumed to take the same value as in the positive direction. Error extents must be positive; negative array elements are ignored.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`xsN = <array-expr>` ***(String)***
Array giving the X coordinate array for each line. In most cases, if a blank value is supplied but Y values are present then a suitable linear sequence, of the same length as the Y array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`yerrhisN = <array-expr>` ***(String)***
Array of errors in the Y coordinates in the positive direction. If no corresponding negative value is supplied, then this value is also used in the negative direction, i.e. in that case errors are assumed to be symmetric. Error exents must be positive; negative array elements are ignored.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`yerrlosN = <array-expr>` ***(String)***
Array of errors in the Y coordinates in the negative direction. If left blank, it is assumed to take the same value as in the positive direction. Error extents must be positive; negative array elements are ignored.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`ysN = <array-expr>` ***(String)***
Array giving the Y coordinate array for each line. In most cases, if a blank value is supplied but X values are present then a suitable linear sequence, of the same length as the X array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

8.3.19 `statline`

Plots a single line based on a combination (typically the mean) of input array-valued coordinates. The input X and Y coordinates must be fixed-length arrays of length N; a line with N points is plotted, each point representing the mean (or median, minimum, maximum, ...) of all the input array elements at the corresponding position.

Note that because the X and Y arrays must be of a fixed size for all rows, and because combination is performed in both X and Y directions, this is typically only suitable for plotting combined spectra

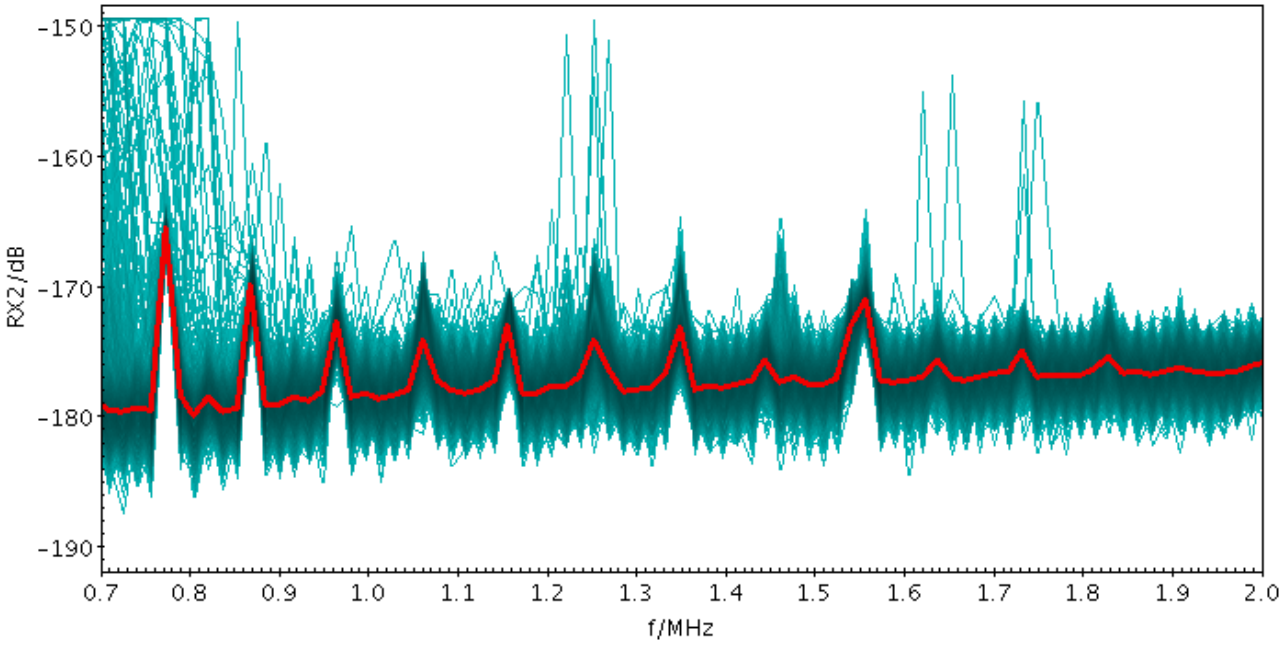
if they all share a common horizontal axis, e.g. are all sampled into the same wavelength bins. To visually combine spectra with non-uniform sampling, the arrayquantile plotter may be more useful.

Usage Overview:

```
layerN=statline xcombineN=mean|median|min|max|q.01|...
                ycombineN=mean|median|min|max|q.01|...
                colorN=<rrggbb>|red|blue|... thickN=<pixels>
                antialiasN=true|false xsN=<array-expr> ysN=<array-expr>
                inN=<table> ifmtN=<in-format> istreamN=true|false
                icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in=LRS_NPW_V010_20071101.cdf xs=multiply(param$frequency,1e-6) ys=RX2
                  xlabel=f/MHz ylabel=RX2/dB xmin=0.7 xmax=2.0 icmd='select rx2[71]<-170' \
                  layer1=lines color1=cyan
                  layer2=statline color2=red thick2=3
```

`antialiasN = true|false` ***(Boolean)***

If true, plotted lines are drawn with antialising. Antialised lines look smoother, but may take perceptibly longer to draw. Only has any effect for bitmapped output formats.

[Default: `false`]

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: red]

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter inN. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter inN. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value (auto) (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: (auto)]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "-", meaning standard input. In this case the input format must be given explicitly using the ifmtN parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form :<scheme-name>:<scheme-args>.
- A system command line with either a "<" character at the start, or a "|" character at the end ("<syscmd" or "syscmd|"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the inN parameter will be read as a stream. It is necessary to give the ifmtN parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: false]

thickN = <pixels> ***(Integer)***

Thickness of plotted line in pixels.

[Default: 3]

xcombineN = mean|median|min|max|q.01|... ***(Combiner)***

Defines how corresponding array elements on the X axis are combined together to produce the plotted value.

The available options are:

- `mean`: the mean of the combined values
- `median`: the median
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `q.01`: 1st percentile
- `q1`: first quartile
- `q3`: third quartile
- `q.99`: 99th percentile
- `stdev`: the sample standard deviation of the combined values
- `sum`: the sum of all the combined values per bin
- `count`: the number of non-blank values per bin (weight is ignored)

[Default: `mean`]

`xsN = <array-expr>` ***(String)***

Array giving the X coordinate array for each line. In most cases, if a blank value is supplied but Y values are present then a suitable linear sequence, of the same length as the Y array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`ycombineN = mean|median|min|max|q.01|...` ***(Combiner)***

Defines how corresponding array elements on the Y axis are combined together to produce the plotted value.

The available options are:

- `mean`: the mean of the combined values
- `median`: the median
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `q.01`: 1st percentile
- `q1`: first quartile
- `q3`: third quartile
- `q.99`: 99th percentile
- `stdev`: the sample standard deviation of the combined values
- `sum`: the sum of all the combined values per bin
- `count`: the number of non-blank values per bin (weight is ignored)

[Default: `mean`]

`ysN = <array-expr>` ***(String)***

Array giving the Y coordinate array for each line. In most cases, if a blank value is supplied but X values are present then a suitable linear sequence, of the same length as the X array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

8.3.20 `statmark`

Plots a set of markers based on a combination (typically the mean) of input array-valued coordinates. The input X and Y coordinates must be fixed-length arrays of length N; N markers are plotted, each one representing the mean (or median, minimum, maximum, ...) of all the input array elements at the corresponding position.

Note that because the X and Y arrays must be of a fixed size for all rows, and because combination

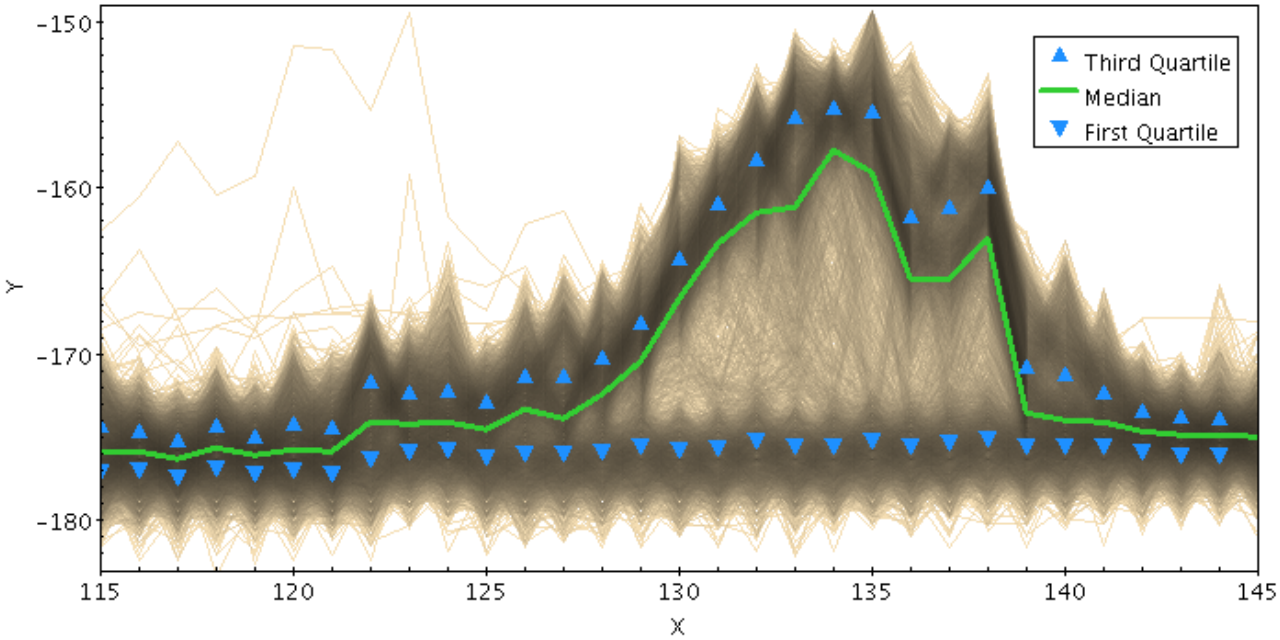
is performed in both X and Y directions, this is typically only suitable for plotting combined spectra if they all share a common horizontal axis, e.g. are all sampled into the same wavelength bins. To visually combine spectra with non-uniform sampling, the arrayquantile plotter may be more useful.

Usage Overview:

```
layerN=statmark xcombineN=mean|median|min|max|q.01|...
                ycombineN=mean|median|min|max|q.01|...
                colorN=<rrggbb>|red|blue|...
                shapeN=filled_circle|open_circle|... sizeN=<pixels>
                xsN=<array-expr> ysN=<array-expr> inN=<table>
                ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in=LRS_NPW_V010_20071101.cdf ys=RX1
                  xmin=115 xmax=145 ymin=-183 ymax=-149 xpix=700 xcrowd=0.8
                  layer-d=lines color-d=wheat
                  layer-m=statline ycombine-m=median color-m=LimeGreen thick-m=3
                  color-q=DodgerBlue size-q=4
                  layer-q1=statmark ycombine-q1=Q1 shape-q1=filled_triangle_up
                  layer-q3=statmark ycombine-q3=Q3 shape-q3=filled_triangle_down
                  leglabel-m=Median leglabel-q1='First Quartile' leglabel-q3='Third Quartil
                  legseq=-q3,-m,-q1 legpos=0.98,0.93
```

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

shapeN = filled_circle|open_circle|... ***(MarkerShape)***

Sets the shape of markers that are plotted at each position of the scatter plot.

The available options are:

- `filled_circle`
- `open_circle`
- `cross`
- `x`
- `open_square`
- `open_diamond`

- `open_triangle_up`
- `open_triangle_down`
- `fat_circle`
- `fat_cross`
- `fat_x`
- `fat_square`
- `fat_diamond`
- `fat_triangle_up`
- `fat_triangle_up`
- `filled_square`
- `filled_diamond`
- `filled_triangle_up`
- `filled_triangle_down`

[Default: `filled_circle`]

`sizeN = <pixels>` ***(Integer)***
Size of the markers. The unit is pixels, in most cases the marker is approximately twice the size of the supplied value.

[Default: `4`]

`xcombineN = mean|median|min|max|q.01|...` ***(Combiner)***
Defines how corresponding array elements on the X axis are combined together to produce the plotted value.

The available options are:

- `mean`: the mean of the combined values
- `median`: the median
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `q.01`: 1st percentile
- `q1`: first quartile
- `q3`: third quartile
- `q.99`: 99th percentile
- `stdev`: the sample standard deviation of the combined values
- `sum`: the sum of all the combined values per bin
- `count`: the number of non-blank values per bin (weight is ignored)

[Default: `mean`]

`xsN = <array-expr>` ***(String)***
Array giving the X coordinate array for each line. In most cases, if a blank value is supplied but Y values are present then a suitable linear sequence, of the same length as the Y array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`ycombineN = mean|median|min|max|q.01|...` ***(Combiner)***
Defines how corresponding array elements on the Y axis are combined together to produce the plotted value.

The available options are:

- `mean`: the mean of the combined values
- `median`: the median
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `q.01`: 1st percentile

- `q1`: first quartile
- `q3`: third quartile
- `q.99`: 99th percentile
- `stdev`: the sample standard deviation of the combined values
- `sum`: the sum of all the combined values per bin
- `count`: the number of non-blank values per bin (weight is ignored)

[Default: `mean`]

`ysN = <array-expr>` ***(String)***

Array giving the Y coordinate array for each line. In most cases, if a blank value is supplied but X values are present then a suitable linear sequence, of the same length as the X array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

8.3.21 `arrayquantile`

Displays a quantile or quantile range for a set of plotted X/Y array pairs. If a table contains one spectrum per row in array-valued wavelength and flux columns, this plotter can be used to display a median of all the spectra, or a range between two quantiles. Smoothing options are available to even out noise arising from the pixel binning.

For each row, the `xs` and `ys` arrays must be the same length as each other, but this plot type does not require all the arrays to be sampled into the same bins.

The algorithm calculates quantiles for all the X,Y points plotted in each column of pixels. This means that more densely sampled spectra have more influence on the output than sparser ones.

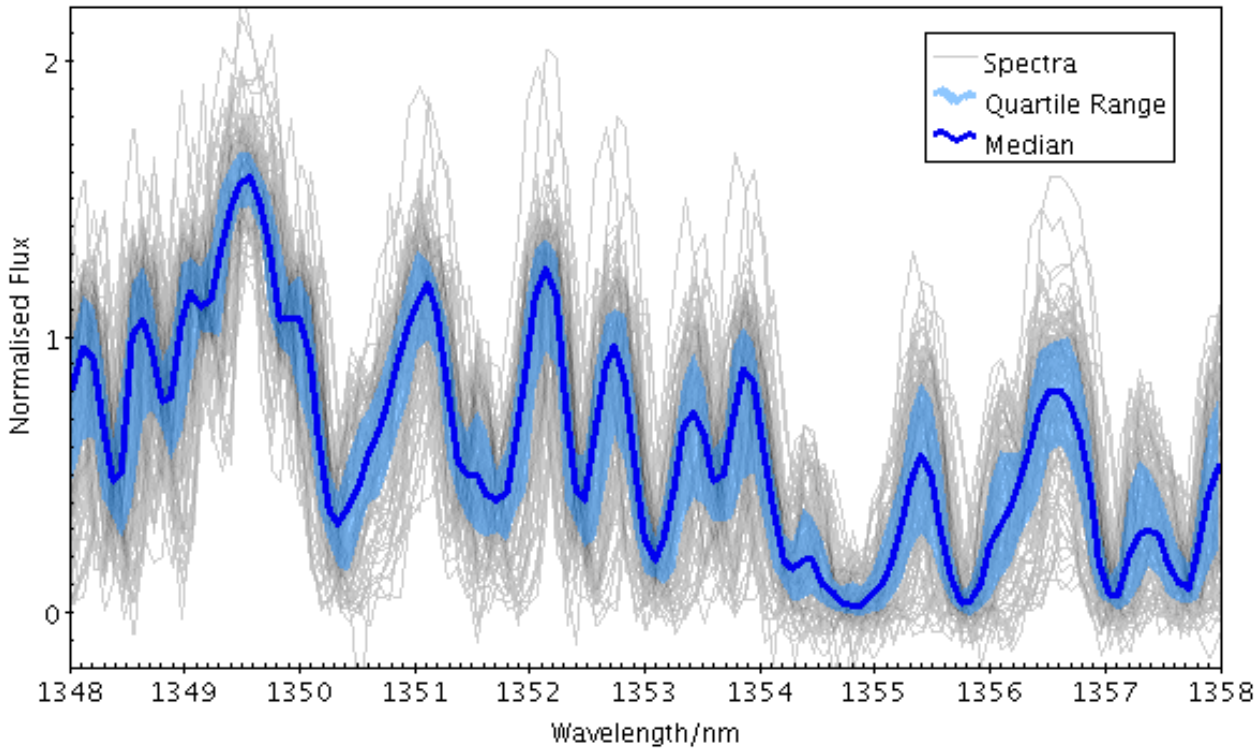
Note: in the current implementation, depending on the details of the configuration and the data, there may be some distortions or missing graphics near the edges of the plot. This may be improved in future releases, depending on feedback.

Usage Overview:

```
layerN=arrayquantile colorN=<rrggbb>|red|blue|... transparencyN=0..1
                     quantilesN=<low-frac>[,<high-frac>] thickN=<pixels>
                     smoothN=+<width>|-<count>
                     kernelN=square|linear|epanechnikov|cos|cos2|gauss3|gauss6
                     joinN=none|polygon|lines horizontalN=true|false
                     xsN=<array-expr> ysN=<array-expr> inN=<table>
                     ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in=xq100sub.fits xs=subWave ys=multiply(subFlux,1./mean(subFlux))
                  xlabel=Wavelength/nm ylabel='Normalised Flux'
                  layer1=lines shading1=density densemap1=greyscale
                  denseclip1=0.2,1 densefunc1=linear leglabel1=Spectra
                  layer_q13=ArrayQuantile color_q13=DodgerBlue transparency_q13=0.5
                  quantiles_q13=0.25,0.75 leglabel_q13='Quartile Range'
                  layer_med=ArrayQuantile color_med=blue join_med=lines leglabel_med=Median
                  legend=true legpos=0.95,0.95
                  xpix=600 ypix=350
                  xmin=1348 xmax=1358 ymin=-0.2 ymax=2.2
```

colorN = <rrggbb>|red|blue|... ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

horizontalN = true|false ***(Boolean)***

Determines whether the trace bins are horizontal or vertical. If `true`, *y* quantiles are calculated for each pixel column, and if `false`, *x* quantiles are calculated for each pixel row.

[Default: `true`]

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is

performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`joinN = none|polygon|lines` ***(QJoin)***

Defines the graphical style for connecting distinct quantile values. If smoothed samples are packed more closely than the pixel grid the option chosen here doesn't make much difference, but if there are gaps in the data along the sampled axis, it's useful to have a guide to the eye to join one quantile determination to the next.

The available options are:

- `none`: displayed quantile ranges are not joined
- `polygon`: the area between a line connecting the upper quantiles and a line connecting the lower quantiles is filled
- `lines`: a line of thickness given by `thick` is drawn from the center of each quantile range to the next

[Default: `polygon`]

`kernelN = square|linear|epanechnikov|cos|cos2|gauss3|gauss6` ***(Kernel1dShape)***

The functional form of the smoothing kernel. The functions listed refer to the unscaled shape;

all kernels are normalised to give a total area of unity.

The available options are:

- `square`: Uniform value: f(x)=1, |x|=0..1
- `linear`: Triangle: f(x)=1-|x|, |x|=0..1
- `epanechnikov`: Parabola: f(x)=1-x*x, |x|=0..1
- `cos`: Cosine: f(x)=cos(x*pi/2), |x|=0..1
- `cos2`: Cosine squared: f(x)=cos^2(x*pi/2), |x|=0..1
- `gauss3`: Gaussian truncated at 3.0 sigma: f(x)=exp(-x*x/2), |x|=0..3
- `gauss6`: Gaussian truncated at 6.0 sigma: f(x)=exp(-x*x/2), |x|=0..6

[Default: `epanechnikov`]

`quantilesN = <low-frac>[,<high-frac>]` ***(Subrange)***

Defines the quantile or quantile range of values that should be marked in each pixel column (or row). The value may be a single number in the range 0..1 indicating the quantile which should be marked. Alternatively, it may be a pair of numbers, each in the range 0..1, separated by commas (`<lo>,<hi>`) indicating two quantile lines bounding an area to be filled. A pair of equal values "`a,a`" is equivalent to the single value "`a`". The default is `0.5`, which means to mark the median value in each column, and could equivalently be specified `0.5,0.5`.

[Default: `0.5`]

`smoothN = +<width>|-<count>` ***(BinSizer)***

Configures the smoothing width. This is the characteristic width of the kernel function to be convolved with the density in one dimension to smooth the quantile function.

If the supplied value is a positive number it is interpreted as a fixed width in the data coordinates of the X axis (if the X axis is logarithmic, the value is a fixed factor). If it is a negative number, then it will be interpreted as the approximate number of smooothing widths that fit in the width of the visible plot (i.e. plot width / smoothing width). If the value is zero, no smoothing is applied.

When setting this value graphically, you can use either the slider to adjust the bin count or the numeric entry field to fix the bin width.

[Default: `0`]

`thickN = <pixels>` ***(Integer)***

Sets the minimum extent of the markers that are plotted in each pixel column (or row) to indicate the designated value range. If the range is zero sized (`quantiles` specifies a single value rather than a pair) this will give the actual thickness of the plotted line. If the range is non-zero however, the line may be thicker than this in places according to the quantile positions.

[Default: `3`]

`transparencyN = 0..1` ***(Double)***

Transparency with which components are plotted, in the range 0 (opaque) to 1 (invisible). The value is 1-alpha.

[Default: `0`]

`xsN = <array-expr>` ***(String)***

Array giving the X coordinate array for each line. In most cases, if a blank value is supplied but Y values are present then a suitable linear sequence, of the same length as the Y array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`ysN = <array-expr>` ***(String)***

Array giving the Y coordinate array for each line. In most cases, if a blank value is supplied but X values are present then a suitable linear sequence, of the same length as the X array, is assumed.

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

8.3.22 `line`

Plots a point-to-point line joining up the positions of data points. There are additional options to pre-sort the points according to their order on the X or Y axis (using the `sortaxis` value), and to vary the colour of the line along its length (using the `aux` value).

The options for controlling the Aux colour map are controlled at the level of the plot itself, rather than by per-layer configuration.

Usage Overview:

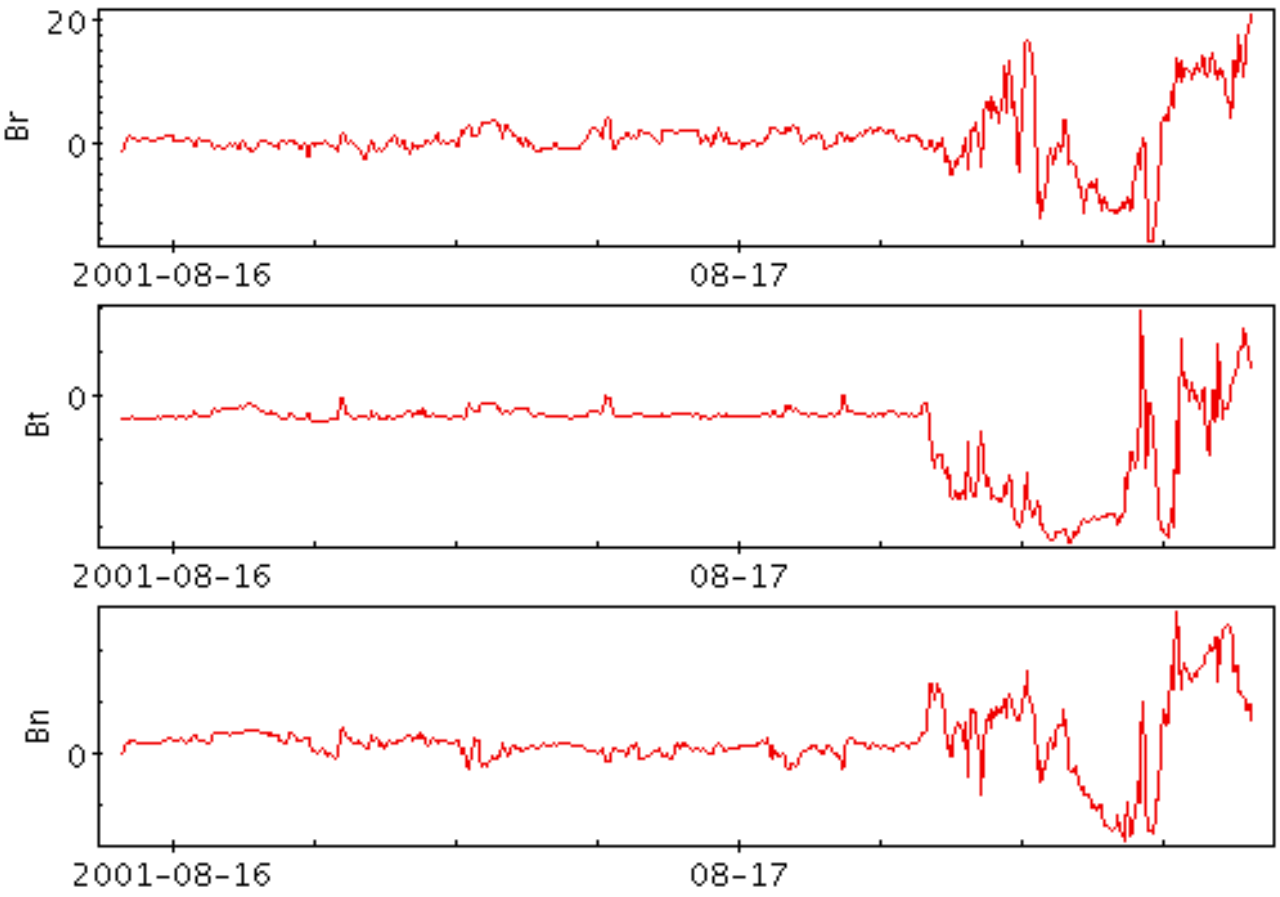
```
layerN=line colorN=<rrggbb>|red|blue|... thickN=<pixels>
            dashN=dot|dash|...|<a,b,...> sortaxisN=[X|Y]
            antialiasN=true|false auxnullcolorN=<rrggbb>|red|blue|...
            <pos-coord-paramsN> auxN=<num-expr> inN=<table>
            ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:

The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2time in=ACE_data.vot t=epoch
                layer1=line y1=Br zone1=A
                layer2=line y2=Bt zone2=B
                layer3=line y3=Bn zone3=C
```

`antialiasN = true|false` ***(Boolean)***

If true, plotted lines are drawn with antialising. Antialised lines look smoother, but may take perceptibly longer to draw. Only has any effect for bitmapped output formats.

[Default: `false`]

`auxN = <num-expr>` ***(String)***

If supplied, this controls the colouring of the line along its length according to the value of this coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`auxnullcolorN = <rrggbb>|red|blue|...` ***(Color)***

The color of points with a null value of the Aux coordinate, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

If the value is null, then points with a null Aux value will not be plotted at all.

[Default: grey]

colorN = <rrggbb>|red|blue|... ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are red, blue, green, grey, magenta, cyan, orange, pink, yellow, black, light_grey, white. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from AliceBlue to YellowGreen, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "#" or "0x", giving red, green and blue intensities, e.g. "ff00ff", "#ff00ff" or "0xff00ff" for magenta.

[Default: red]

dashN = dot|dash|...|<a,b,...> ***(float[])***

Determines the dash pattern of the line drawn. If null (the default), the line is solid.

Possible values for dashed lines are dot, dash, longdash, dotdash. You can alternatively supply a comma-separated list of on/off length values such as "4,2,8,2".

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter inN. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter inN. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value (auto) (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: (auto)]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "-", meaning standard input. In this case the input format must be given explicitly using the ifmtN parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form :<scheme-name>:<scheme-args>.
- A system command line with either a "<" character at the start, or a "|" character at the end ("<syscmd" or "syscmd|"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`sortaxisN = [X|Y]` ***(AxisOpt)***

May be set to "`X`" or "`Y`" to ensure that the points are plotted in ascending order of the corresponding coordinate. This will ensure that the plotted line resembles a function of the corresponding coordinate rather than a scribble. The default (null) value causes the points to be joined in the sequence in which they appear in the table. If the points already appear in the table sorted according to the corresponding coordinate, this option has no visible effect, though it may slow things down.

[Default: `None`]

`thickN = <pixels>` ***(Integer)***

Thickness of plotted line in pixels.

[Default: `1`]

8.3.23 `linearfit`

Plots a line of best fit for the data points.

Usage Overview:

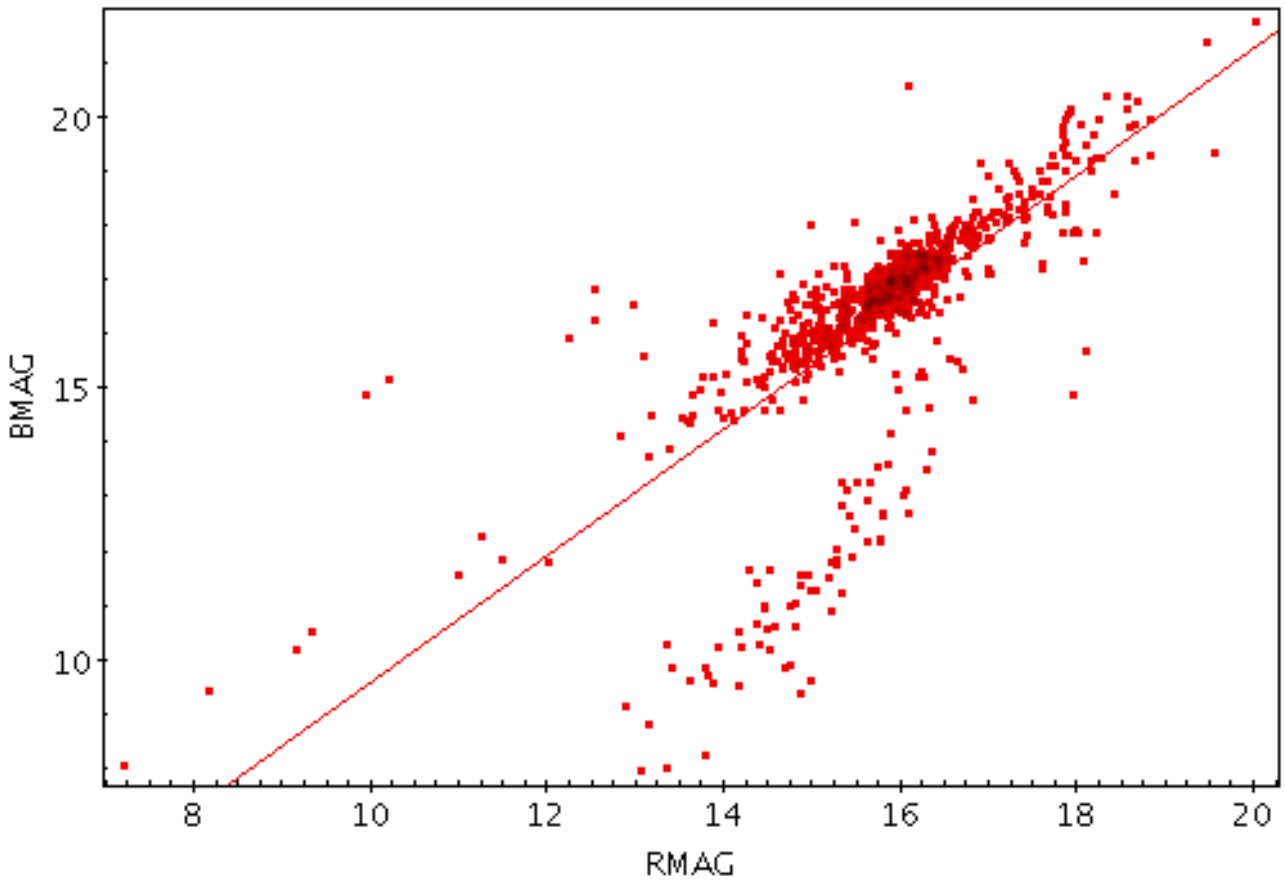
```
layerN=linearfit colorN=<rrggbb>|red|blue|... thickN=<pixels>
                 dashN=dot|dash|...|<a,b,...> antialiasN=true|false
                 <pos-coord-paramsN> weightN=<num-expr> inN=<table>
                 ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:

The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2plane in=6dfgs_mini.xml x=RMAG y=BMAG layer1=mark layer2=linearfit
```

`antialiasN = true|false` ***(Boolean)***

If true, plotted lines are drawn with antialising. Antialised lines look smoother, but may take perceptibly longer to draw. Only has any effect for bitmapped output formats.

[Default: `false`]

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`dashN = dot|dash|...|<a,b,...>` ***(float[])***

Determines the dash pattern of the line drawn. If null (the default), the line is solid.

Possible values for dashed lines are `dot`, `dash`, `longdash`, `dotdash`. You can alternatively supply a comma-separated list of on/off length values such as "`4,2,8,2`".

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If

more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`thickN = <pixels>` ***(Integer)***

Thickness of plotted line in pixels.

[Default: `1`]

`weightN = <num-expr>` ***(String)***

The weight associated with each data point for fitting purposes. This is used for calculating the coefficients of the line of best fit, and the correlation coefficient. If no coordinate is supplied, all points are assumed to have equal weight (1). Otherwise, any point with a null weight value is assigned a weight of zero, i.e. ignored.

Given certain assumptions about independence of samples, a suitable value for the weight may be `1/(err*err)`, if `err` is the measurement error for each Y value.

The value is a numeric algebraic expression based on column names as described in Section

10.

8.3.24 `label`

Draws a text label at each position. You can select the font, where the labels appear in relation to the point positions, and how crowded the points have to get before they are suppressed.

Usage Overview:

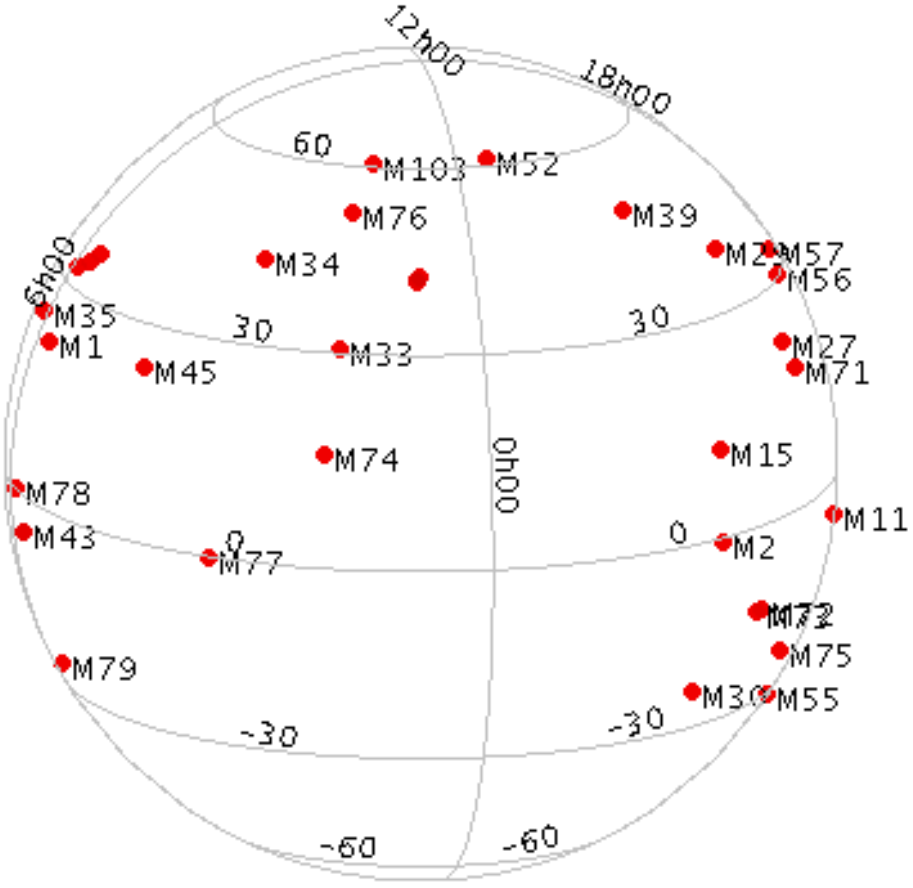
```
layerN=label texttypeN=plain|antialias|latex fontsizeN=<int-value>
             fontstyleN=standard|serif|mono
             fontweightN=plain|bold|italic|bold_italic
             anchorN=west|east|north|south|center
             colorN=<rrggbb>|red|blue|... xoffN=<pixels> yoffN=<pixels>
             spacingN=<pixels> crowdlimitN=<n> <pos-coord-paramsN>
             labelN=<txt-expr> inN=<table> ifmtN=<in-format>
             istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:

The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:



```
stilts plot2sky in=messier.xml lon=RA lat=DEC
               layer1=mark size1=3
               layer2=label label2=NAME color2=black
```

anchorN = west|east|north|south|center ***(Anchor)***

Determines where the text appears in relation to the plotted points. Values are points of the compass.

The available options are:

- `west`
- `east`
- `north`
- `south`
- `center`

[Default: `west`]

colorN = <rrggbb>|red|blue|... ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

crowdlimitN = <n> ***(Integer)***

Sets the maximum number of labels in a label group. This many labels can appear closely spaced without being affected by the label spacing parameter.

It is useful for instance if you are looking at pairs of points, which will always be close together; if you set this value to 2, an isolated pair of labels can be seen, but if it's 1 then they will only be plotted when they are distant from each other, which may only happen at very high magnifications.

[Default: `2`]

fontsizeN = <int-value> ***(Integer)***

Size of the text font in points.

[Default: `12`]

fontstyleN = standard|serif|mono ***(FontType)***

Font style for text.

The available options are:

- `standard`
- `serif`
- `mono`

[Default: `standard`]

fontweightN = plain|bold|italic|bold_italic ***(FontWeight)***

Font weight for text.

The available options are:

- `plain`
- `bold`
- `italic`
- `bold_italic`

[Default: `plain`]

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`labelN = <txt-expr>` ***(String)***
Column or expression giving the text of the label to be written near the position being labelled.

Label values may be of any type (string or numeric)

The value is a string algebraic expression based on column names as described in Section 10.

`spacingN = <pixels>` ***(Integer)***
Determines the closest that labels can be spaced. If a group of labels is closer to another group than the value of this parameter, they will not be drawn, to avoid the display becoming too cluttered. The effect is that you can see individual labels when you zoom in, but not when there are many labelled points plotted close together on the screen. Set the value higher for less cluttered labelling.

[Default: `12`]

`texttypeN = plain|antialias|latex` ***(TextSyntax)***
Determines how to turn label text into characters on the plot. `Plain` and `Antialias` both take the text at face value, but `Antialias` smooths the characters. `LaTeX` interprets the text as LaTeX source code and typesets it accordingly.

When not using LaTeX, antialiased text usually looks nicer, but can be perceptibly slower to plot. At time of writing, on MacOS antialiased text seems to be required to stop the writing coming out upside-down for non-horizontal text (MacOS java bug).

[Default: `plain`]

`xoffN = <pixels>` ***(Integer)***
Allows fine adjustment of label positioning in the X direction. The value is a positive or negative pixel offset applied to the position of each plotted label.

[Default: `0`]

`yoffN = <pixels>` ***(Integer)***
Allows fine adjustment of label positioning in the Y direction. The value is a positive or negative pixel offset applied to the position of each plotted label.

[Default: `0`]

8.3.25 `arealabel`

Draws a text label near the center of each area.You can select the font, where the labels appear in relation to the point positions, and how crowded the points have to get before they are suppressed.

This is just like a normal Label plot, but the positions are taken from an Area coordinate rather than normal positional coordinates.

Usage Overview:

```
layerN=arealabel texttypeN=plain|antialias|latex fontsizeN=<int-value>
                 fontstyleN=standard|serif|mono
                 fontweightN=plain|bold|italic|bold_italic
                 anchorN=west|east|north|south|center
                 colorN=<rrggbb>|red|blue|... xoffN=<pixels> yoffN=<pixels>
                 spacingN=<pixels> crowdlimitN=<n> areaN=<area-expr>
                 areatypeN=STC-S|POLYGON|CIRCLE|POINT|MOC-ASCII|UNIQ
                 labelN=<txt-expr> inN=<table> ifmtN=<in-format>
                 istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:

```
stilts plot2sky reflectlon=false sex=false
               clon=18 clat=0 radius=36 xpix=550 ypix=600
               in=countries.vot
               area=shape areatype=STC-S
               layer_1=area polymode_1=fill
               shading_1=aux aux_1=index opaque_1=2 layer_2=area polymode_2=outline
               shading_2=flat color_2=grey
               auxmap=paired auxvisible=false
               layer_3=arealabel label_3=name anchor_3=center color_3=black
```

anchorN = west|east|north|south|center ***(Anchor)***

Determines where the text appears in relation to the plotted points. Values are points of the compass.

The available options are:

- west

- east
- north
- south
- center

[Default: west]

areaN = <area-expr> ***(String)***

Expression giving the geometry of a 2D region on the plot. It may be a string- or array-valued expression, and its interpretation depends on the value of the corresponding areatype parameter.

The value is a Area value algebraic expression based on column names as described in Section 10.

areatypeN = STC-S|POLYGON|CIRCLE|POINT|MOC-ASCII|UNIQ ***(AreaMapper)***

Selects the form in which the Area value for parameter areaN is supplied. Options are:

- STC-S: Region description using STC-S syntax; see TAP 1.0, section 6. Note there are some restrictions: <frame>, <refpos> and <flavor> metadata are ignored, polygon winding direction is ignored (small polygons are assumed) and the INTERSECTION and NOT constructions are not supported. The non-standard MOC construction is supported.
- POLYGON: 2n-element array (x1,y1, x2,y2,...); a NaN,NaN pair can be used to delimit distinct polygons.
- CIRCLE: 3-element array (x,y,r)
- POINT: 2-element array (x,y)
- MOC-ASCII: Region description using ASCII MOC syntax; see MOC 1.1 2.3.2. Note there are currently a few issues with MOC plotting, especially for large pixels.
- UNIQ: Region description representing a single HEALPix cell as defined by an UNIQ value, see MOC 1.1 sec 2.3.1.

If left blank, a guess will be taken depending on the data type of the value supplied for the areaN value.

colorN = <rrggbb>|red|blue|... ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are red, blue, green, grey, magenta, cyan, orange, pink, yellow, black, light_grey, white. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from AliceBlue to YellowGreen, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "#" or "0x", giving red, green and blue intensities, e.g. "ff00ff", "#ff00ff" or "0xff00ff" for magenta.

[Default: red]

crowdlimitN = <n> ***(Integer)***

Sets the maximum number of labels in a label group. This many labels can appear closely spaced without being affected by the label spacing parameter.

It is useful for instance if you are looking at pairs of points, which will always be close together; if you set this value to 2, an isolated pair of labels can be seen, but if it's 1 then they will only be plotted when they are distant from each other, which may only happen at very high magnifications.

[Default: 2]

fontsizeN = <int-value> ***(Integer)***

Size of the text font in points.

[Default: 12]

fontstyleN = standard|serif|mono ***(FontType)***

Font style for text.

The available options are:

- standard
- serif
- mono

[Default: standard]

fontweightN = plain|bold|italic|bold_italic ***(FontWeight)***

Font weight for text.

The available options are:

- plain
- bold
- italic
- bold_italic

[Default: plain]

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter inN. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter inN. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value (auto) (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: (auto)]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "-", meaning standard input. In this case the input format must be given explicitly using the ifmtN parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form :<scheme-name>:<scheme-args>.
- A system command line with either a "<" character at the start, or a "|" character at the end ("<syscmd" or "syscmd|"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix

compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`labelN = <txt-expr>` ***(String)***
Column or expression giving the text of the label to be written near the position being labelled. Label values may be of any type (string or numeric)

The value is a string algebraic expression based on column names as described in Section 10.

`spacingN = <pixels>` ***(Integer)***
Determines the closest that labels can be spaced. If a group of labels is closer to another group than the value of this parameter, they will not be drawn, to avoid the display becoming too cluttered. The effect is that you can see individual labels when you zoom in, but not when there are many labelled points plotted close together on the screen. Set the value higher for less cluttered labelling.

[Default: `12`]

`texttypeN = plain|antialias|latex` ***(TextSyntax)***
Determines how to turn label text into characters on the plot. `Plain` and `Antialias` both take the text at face value, but `Antialias` smooths the characters. `LaTeX` interprets the text as LaTeX source code and typesets it accordingly.

When not using LaTeX, antialiased text usually looks nicer, but can be perceptibly slower to plot. At time of writing, on MacOS antialiased text seems to be required to stop the writing coming out upside-down for non-horizontal text (MacOS java bug).

[Default: `plain`]

`xoffN = <pixels>` ***(Integer)***
Allows fine adjustment of label positioning in the X direction. The value is a positive or negative pixel offset applied to the position of each plotted label.

[Default: `0`]

`yoffN = <pixels>` ***(Integer)***
Allows fine adjustment of label positioning in the Y direction. The value is a positive or negative pixel offset applied to the position of each plotted label.

[Default: `0`]

8.3.26 `contour`

Plots position density contours. This provides another way (alongside the auto, density and weighted shading modes) to visualise the characteristics of overdense regions in a crowded plot. It's not very useful if you just have a few points.

A weighting may optionally be applied to the quantity being contoured. To do this, provide a non-blank value for the `weight` coordinate, and use the `combine` parameter to define how the weights are combined (`sum`, `mean`, etc).

The contours are currently drawn as pixels rather than lines so they don't look very beautiful in exported vector output formats (PDF, PostScript). This may be improved in the future.

Usage Overview:

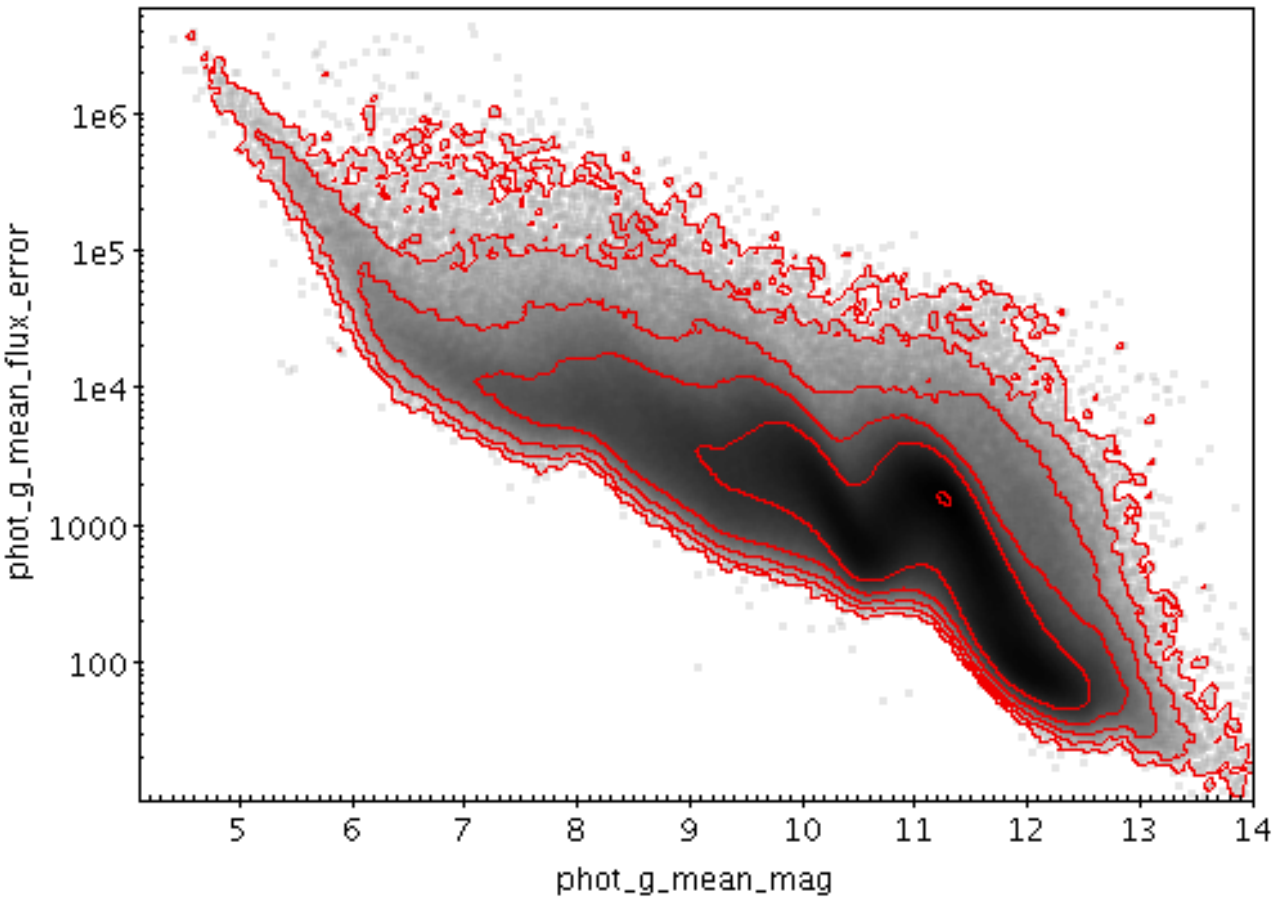
```
layerN=contour colorN=<rrggbb>|red|blue|...
               combineN=sum|mean|median|stdev|min|max|count
               nlevelN=<int-value> smoothN=<pixels> thickN=<pixels>
               scalingN=linear|log|equal zeroN=<number> <pos-coord-paramsN>
               weightN=<num-expr> inN=<table> ifmtN=<in-format>
               istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:

The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2plane in=tgas_source.fits x=phot_g_mean_mag y=phot_g_mean_flux_error
                  ylog=true xmax=14 ymin=10
                  layer1=mark shading1=density densemap1=greyscale
                  layer2=contour scaling2=log nlevel=6
```

colorN = <rrggbb>|red|blue|... ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`combineN = sum|mean|median|stdev|min|max|count` ***(Combiner)***

Defines the way that the weight values are combined when generating the value grid for which the contours will be plotted. If a weighting is supplied, the most useful values are `mean` which traces the mean values of a quantity and `sum` which traces the weighted sum. Other values such as `median` are of dubious validity because of the way that the smoothing is done.

This value is ignored if the weighting coordinate `weight` is not set.

The available options are:

- `sum`: the sum of all the combined values per bin
- `mean`: the mean of the combined values
- `median`: the median
- `stdev`: the sample standard deviation of the combined values
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `count`: the number of non-blank values per bin (weight is ignored)

[Default: `sum`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.

- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`nlevelN = <int-value>` ***(Integer)***
Number of countour lines drawn. In fact, this is an upper limit; if there is not enough variation in the plot's density, then fewer contour lines will be drawn.

[Default: `5`]

`scalingN = linear|log|equal` ***(LevelMode)***
How the smoothed density is treated before contour levels are determined.

The available options are:

- `linear`: levels are equally spaced
- `log`: level logarithms are equally spaced
- `equal`: levels are spaced to provide equal-area inter-contour regions

[Default: `linear`]

`smoothN = <pixels>` ***(Integer)***
The linear size of the smoothing kernel applied to the density before performing the contour determination. If set too low the contours will be too crinkly, and if too high they will lose definition. Smoothing currently uses an approximately Gaussian kernel for extensive combination modes (count, sum) or a circular top hat for intensive modes (weighted mean).

[Default: `5`]

`thickN = <pixels>` ***(Integer)***
Thickness of plotted line in pixels.

[Default: `1`]

`weightN = <num-expr>` ***(String)***
Weighting of data points. If supplied, each point contributes a value to the histogram equal to the data value multiplied by this coordinate. If not supplied, the effect is the same as supplying a fixed value of one.

The value is a numeric algebraic expression based on column names as described in Section 10.

`zeroN = <number>` ***(Double)***
Determines the level at which the first contour (and hence all the others, which are separated from it by a fixed amount) are drawn.

[Default: `1`]

8.3.27 `grid`

Plots 2-d data aggregated into rectangular cells. You can optionally use a weighting for the points, and you can configure how the values are combined to produce the output pixel values (colours). You can use this plotter in various ways, including as a 2-d histogram or weighted density map, or to plot gridded data.

The X and Y dimensions of the grid cells (or equivalently histogram bins) can be configured either in terms of the data coordinates or relative to the plot dimensions.

The way that data values are mapped to colours is usually controlled by options at the level of the plot itself, rather than by per-layer configuration.

Usage Overview:

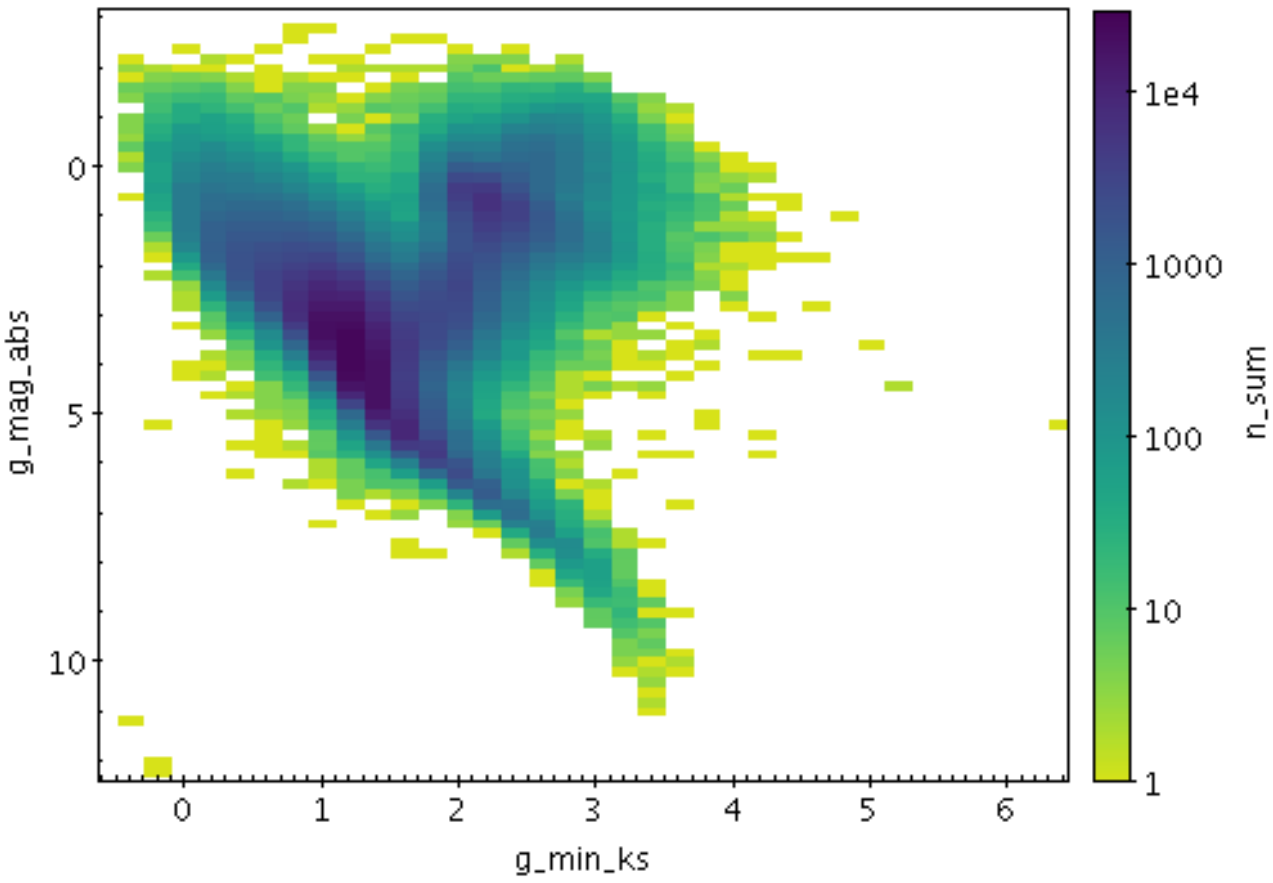
```
layerN=grid xbinsizeN=+<extent>|-<count> ybinsizeN=+<extent>|-<count>
            combineN=sum|sum-per-unit|count|... transparencyN=0..1
            xphaseN=<number> yphaseN=<number> <pos-coord-paramsN>
            weightN=<num-expr> inN=<table> ifmtN=<in-format>
            istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:

The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2plane layer1=grid in1=gk_hess.fits x1=g_min_ks y1=g_mag_abs
                  weight1=n combine1=sum xbinsize1=0.2 ybinsize1=0.2 xphase1=0.5 yphase1=0.
                  yflip=true auxfunc=log auxmap=viridis
```

`combineN = sum|sum-per-unit|count|...` ***(Combiner)***

Defines how values contributing to the same grid cell are combined together to produce the value assigned to that cell, and hence its colour. The combined values are the weights, but if the `weight` coordinate is left blank, a weighting of unity is assumed.

The available options are:

- `sum`: the sum of all the combined values per bin
- `sum-per-unit`: the sum of all the combined values per unit of bin size
- `count`: the number of non-blank values per bin (weight is ignored)
- `count-per-unit`: the number of non-blank values per unit of bin size (weight is ignored)
- `mean`: the mean of the combined values
- `median`: the median
- `q1`: first quartile
- `q3`: third quartile
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `stdev`: the sample standard deviation of the combined values
- `hit`: 1 if any values present, NaN otherwise (weight is ignored)

[Default: `mean`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter

can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`transparencyN = 0..1` ***(Double)***
Transparency with which components are plotted, in the range 0 (opaque) to 1 (invisible). The value is 1-alpha.

[Default: `0`]

`weightN = <num-expr>` ***(String)***
Weighting of data points. If supplied, each point contributes a value to the histogram equal to the data value multiplied by this coordinate. If not supplied, the effect is the same as supplying a fixed value of one.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xbinsizeN = +<extent>|-<count>` ***(BinSizer)***
Configures the extent of the density grid bins on the X axis.

If the supplied value is a positive number it is interpreted as a fixed size in data coordinates (if the X axis is logarithmic, the value is a fixed factor). If it is a negative number, then it will be interpreted as the approximate number of bins to display across the plot in the X direction (though an attempt is made to use only round numbers for bin sizes).

When setting this value graphically, you can use either the slider to adjust the bin count or the numeric entry field to fix the bin size.

[Default: `-30`]

`xphaseN = <number>` ***(Double)***

Controls where the zero point on the X axis is set. For instance if your bin size is 1, this value controls whether bin boundaries are at 0, 1, 2, .. or 0.5, 1.5, 2.5, ... etc.

A value of 0 (or any integer) will result in a bin boundary at X=0 (linear X axis) or X=1 (logarithmic X axis). A fractional value will give a bin boundary at that value multiplied by the bin width.

[Default: `0`]

`ybinsizeN = +<extent>|-<count>` ***(BinSizer)***

Configures the extent of the density grid bins on the Y axis.

If the supplied value is a positive number it is interpreted as a fixed size in data coordinates (if the Y axis is logarithmic, the value is a fixed factor). If it is a negative number, then it will be interpreted as the approximate number of bins to display across the plot in the Y direction (though an attempt is made to use only round numbers for bin sizes).

When setting this value graphically, you can use either the slider to adjust the bin count or the numeric entry field to fix the bin size.

[Default: `-30`]

`yphaseN = <number>` ***(Double)***

Controls where the zero point on the Y axis is set. For instance if your bin size is 1, this value controls whether bin boundaries are at 0, 1, 2, .. or 0.5, 1.5, 2.5, ... etc.

A value of 0 (or any integer) will result in a bin boundary at X=0 (linear X axis) or X=1 (logarithmic X axis). A fractional value will give a bin boundary at that value multiplied by the bin width.

[Default: `0`]

8.3.28 `fill`

If a two-dimensional dataset represents a single-valued function, this plotter will fill the area underneath the function's curve with a solid colour. Parts of the surface which would only be partially covered (because of rapid function variation within the width of a single pixel) are represented using appropriate alpha-blending. The filled area may alternatively be that above the curve or to its left or right.

One example of its use is to reconstruct the appearance of a histogram plot from a set of histogram bins. For X,Y data which is not single-valued, the result may not be very useful.

Usage Overview:

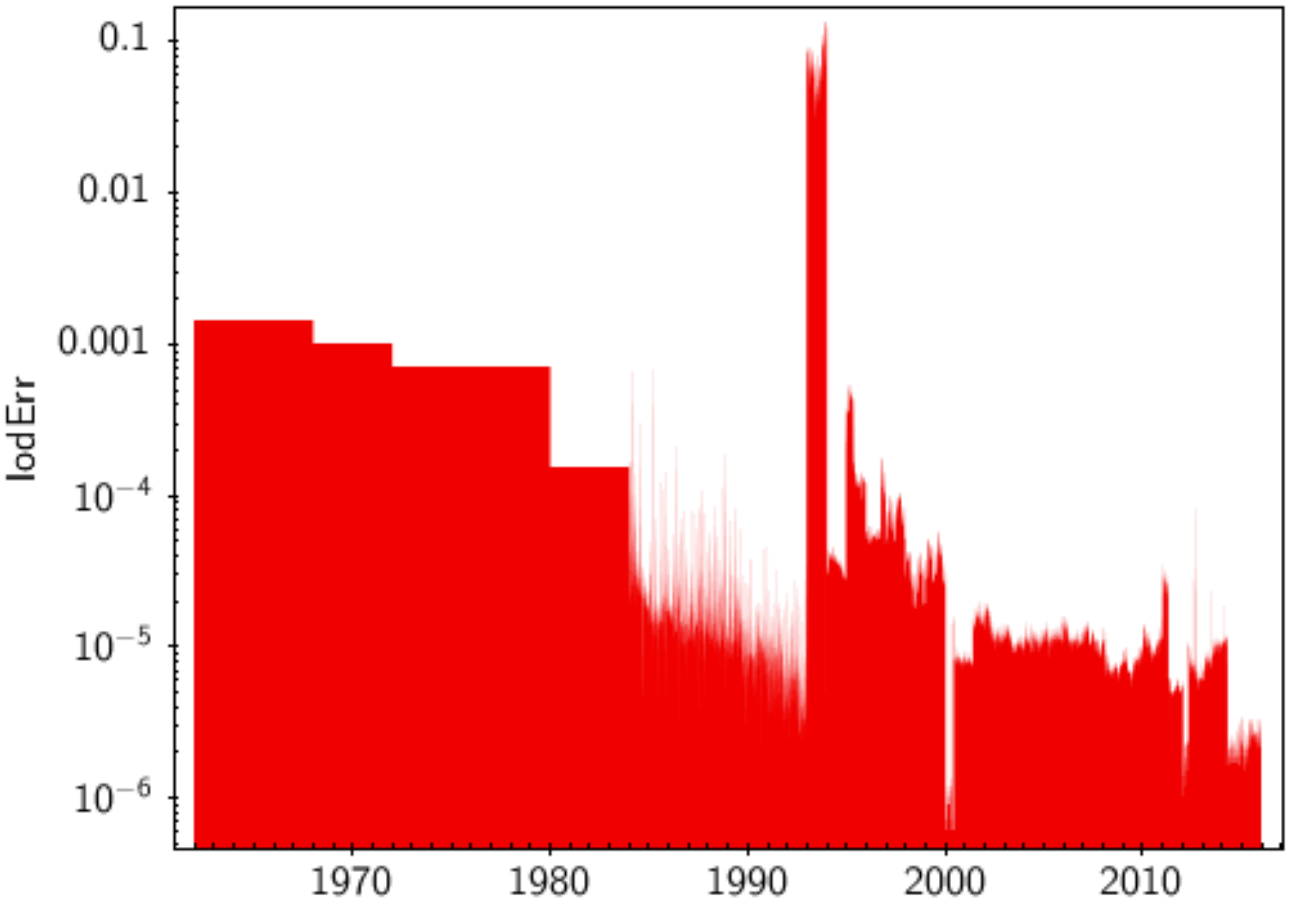
```
layerN=fill colorN=<rrggbb>|red|blue|... transparencyN=0..1
            horizontalN=true|false positiveN=true|false <pos-coord-paramsN>
            inN=<table> ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:

The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2time layer1=fill in1=iers.fits t1=decYear y1=lodErr ylog=true
                 texttype=latex fontsize=16
```

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`horizontalN = true|false` ***(Boolean)***

Determines whether the filling is vertical (suitable for functions of the horizontal variable), or

horizontal (suitable for functions of the vertical variable). If false, the fill is vertical (to the X axis), and if true, the fill is horizontal (to the Y axis).

[Default: `false`]

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`positiveN = true|false` ***(Boolean)***
Determines the directional sense of the filling. If false, the fill is between the data points and negative infinity along the relevant axis (e.g. down from the data points to the bottom of the plot). If true, the fill is in the other direction.

[Default: `false`]

`transparencyN = 0..1` ***(Double)***

Transparency with which components are plotted, in the range 0 (opaque) to 1 (invisible). The value is 1-alpha.

[Default: `0`]

8.3.29 `quantile`

Plots a line through a given quantile of the values binned within each pixel column (or row) of a plot. The line is optionally smoothed using a configurable kernel and width, to even out noise arising from the pixel binning. Instead of a simple line through a given quantile, it is also possible to fill the region between two quantiles.

One way to use this is to draw a line estimating a function *y=f(x)* (or *x=g(y)*) sampled by a noisy set of data points in two dimensions.

Note: in the current implementation, depending on the details of the configuration and the data, there may be some distortions or missing graphics near the edges of the plot. This may be improved in future releases, depending on feedback.

Usage Overview:

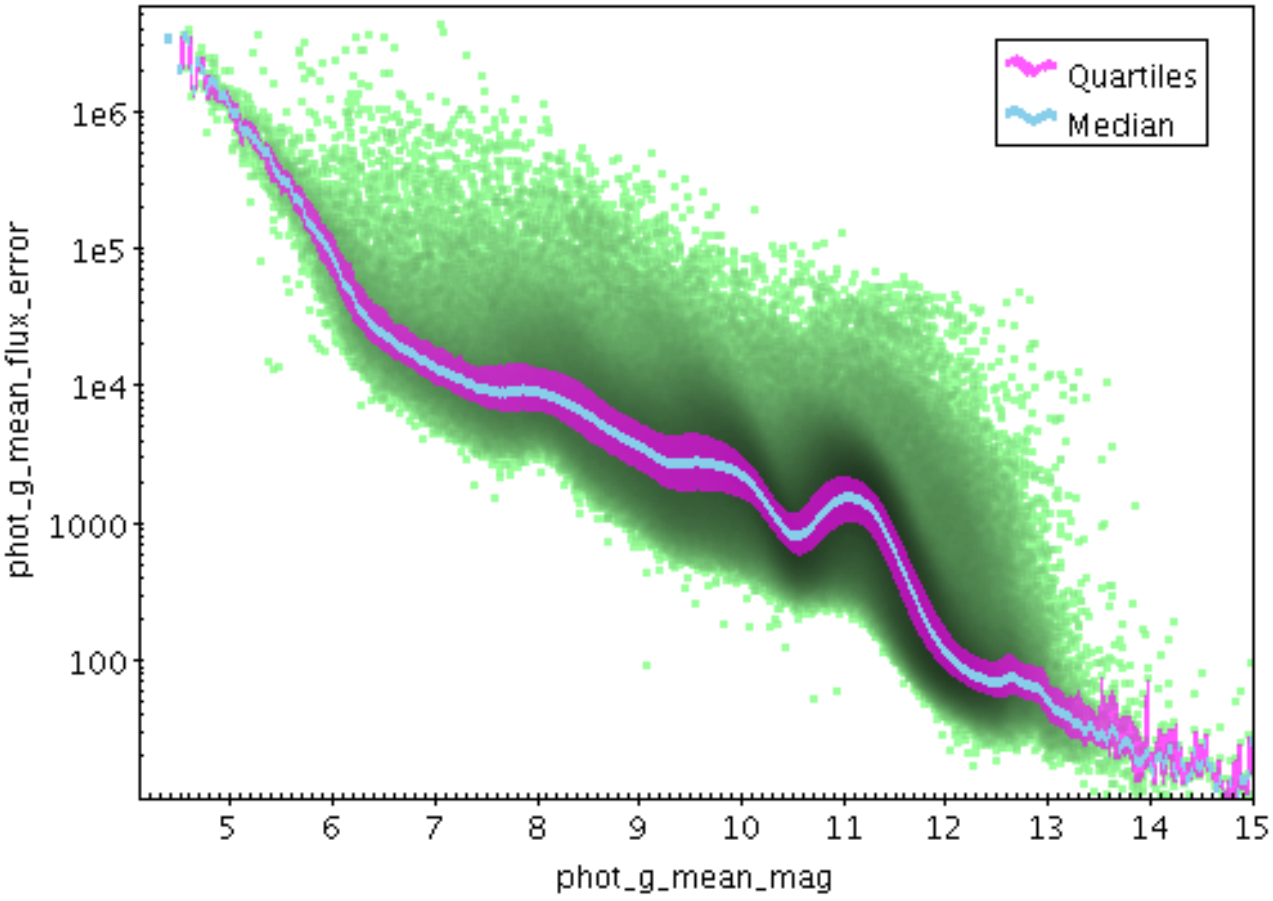
```
layerN=quantile colorN=<rrggbb>|red|blue|... transparencyN=0..1
                quantilesN=<low-frac>[,<high-frac>] thickN=<pixels>
                smoothN=+<width>|-<count>
                kernelN=square|linear|epanechnikov|cos|cos2|gauss3|gauss6
                joinN=none|polygon|lines horizontalN=true|false
                <pos-coord-paramsN> inN=<table> ifmtN=<in-format>
                istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:

The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2plane in=tgas_source.fits x=phot_g_mean_mag y=phot_g_mean_flux_error
                  ylog=true xmax=15 ymin=10
                  layer.d=mark color.d=99ff99
                  layer.q4=quantile quantiles.q4=0.25,0.75 color.q4=magenta transparency.q4
                  layer.q2=quantile quantiles.q2=0.5 color.q2=SkyBlue thick.q2=4
                  smooth.q=0.05
                  leglabel.q4=Quartiles leglabel.q2=Median legseq=.q4,.q2 legpos=0.95,0.95
```

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`horizontalN = true|false` ***(Boolean)***

Determines whether the trace bins are horizontal or vertical. If `true`, *y* quantiles are calculated for each pixel column, and if `false`, *x* quantiles are calculated for each pixel row.

[Default: `true`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter

can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`joinN = none|polygon|lines` ***(QJoin)***

Defines the graphical style for connecting distinct quantile values. If smoothed samples are packed more closely than the pixel grid the option chosen here doesn't make much difference, but if there are gaps in the data along the sampled axis, it's useful to have a guide to the eye to join one quantile determination to the next.

The available options are:

- `none`: displayed quantile ranges are not joined
- `polygon`: the area between a line connecting the upper quantiles and a line connecting the lower quantiles is filled
- `lines`: a line of thickness given by `thick` is drawn from the center of each quantile range to the next

[Default: `none`]

`kernelN = square|linear|epanechnikov|cos|cos2|gauss3|gauss6` ***(Kernel1dShape)***
The functional form of the smoothing kernel. The functions listed refer to the unscaled shape; all kernels are normalised to give a total area of unity.

The available options are:

- `square`: Uniform value: f(x)=1, |x|=0..1
- `linear`: Triangle: f(x)=1-|x|, |x|=0..1
- `epanechnikov`: Parabola: f(x)=1-x*x, |x|=0..1
- `cos`: Cosine: f(x)=cos(x*pi/2), |x|=0..1
- `cos2`: Cosine squared: f(x)=cos^2(x*pi/2), |x|=0..1
- `gauss3`: Gaussian truncated at 3.0 sigma: f(x)=exp(-x*x/2), |x|=0..3
- `gauss6`: Gaussian truncated at 6.0 sigma: f(x)=exp(-x*x/2), |x|=0..6

[Default: `epanechnikov`]

`quantilesN = <low-frac>[,<high-frac>]` ***(Subrange)***
Defines the quantile or quantile range of values that should be marked in each pixel column (or row). The value may be a single number in the range 0..1 indicating the quantile which should be marked. Alternatively, it may be a pair of numbers, each in the range 0..1, separated by commas (`<lo>,<hi>`) indicating two quantile lines bounding an area to be filled. A pair of equal values "`a,a`" is equivalent to the single value "`a`". The default is `0.5`, which means to mark the median value in each column, and could equivalently be specified `0.5,0.5`.

[Default: `0.5`]

`smoothN = +<width>|-<count>` ***(BinSizer)***
Configures the smoothing width. This is the characteristic width of the kernel function to be convolved with the density in one dimension to smooth the quantile function.

If the supplied value is a positive number it is interpreted as a fixed width in the data coordinates of the X axis (if the X axis is logarithmic, the value is a fixed factor). If it is a negative number, then it will be interpreted as the approximate number of smooothing widths that fit in the width of the visible plot (i.e. plot width / smoothing width). If the value is zero, no smoothing is applied.

When setting this value graphically, you can use either the slider to adjust the bin count or the numeric entry field to fix the bin width.

[Default: `0`]

`thickN = <pixels>` ***(Integer)***
Sets the minimum extent of the markers that are plotted in each pixel column (or row) to indicate the designated value range. If the range is zero sized (`quantiles` specifies a single value rather than a pair) this will give the actual thickness of the plotted line. If the range is non-zero however, the line may be thicker than this in places according to the quantile positions.

[Default: `3`]

`transparencyN = 0..1` ***(Double)***
Transparency with which components are plotted, in the range 0 (opaque) to 1 (invisible). The value is 1-alpha.

[Default: `0`]

8.3.30 `histogram`

Plots a histogram.

Bin heights may optionally be weighted by the values of some additional coordinate, supplied using

the `weight` parameter. In this case you can choose how these weights are combined in each bin using the `combine` parameter.

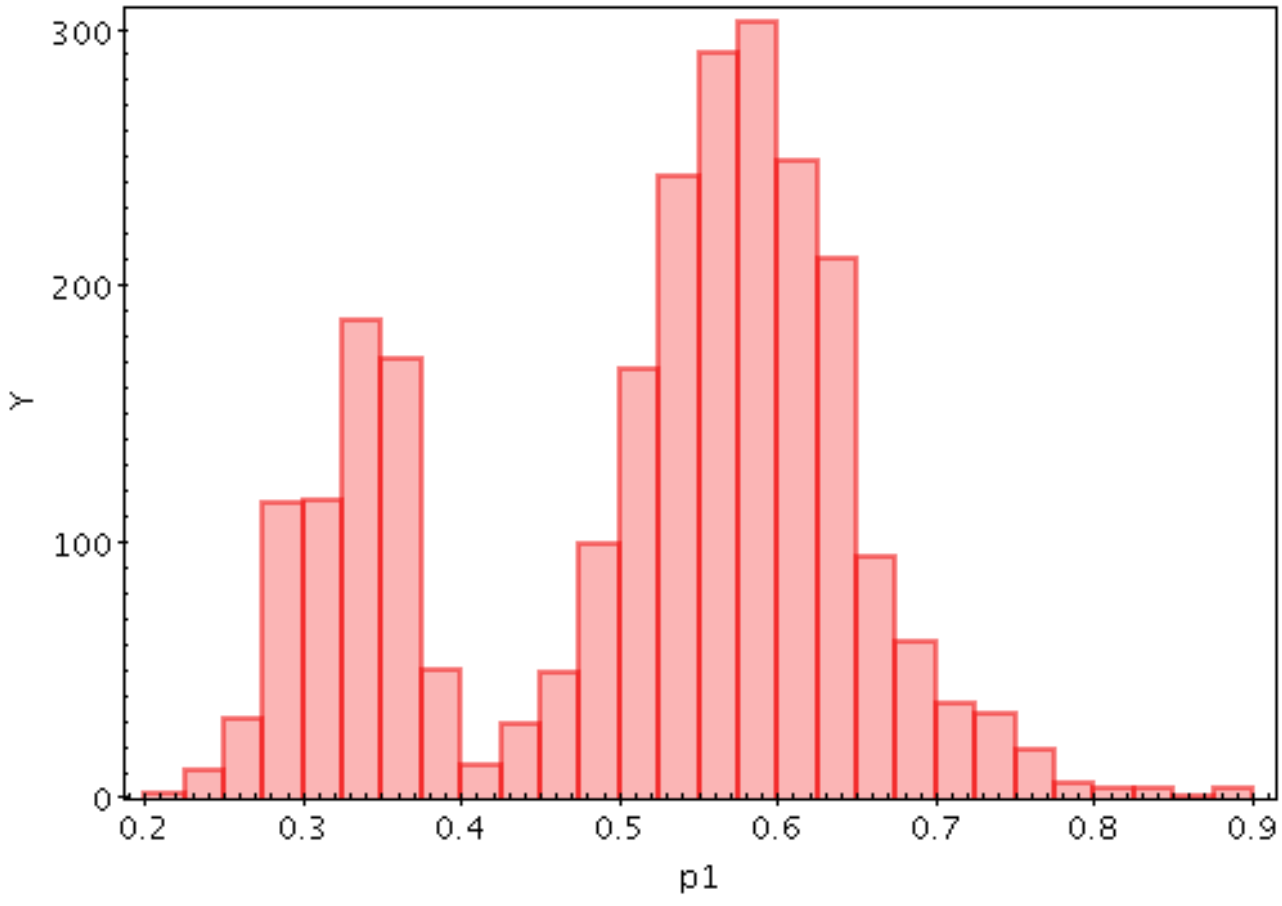
Various options are provided for configuring how the bar heights are calculated, but note that not all combinations of the available parameters will necessarily lead to meaningful visualisations.

Usage Overview:

```
layerN=histogram colorN=<rrggbb>|red|blue|... transparencyN=0..1
                 binsizeN=+<width>|-<count> phaseN=<number>
                 combineN=sum|sum-per-unit|count|...
                 cumulativeN=none|forward|reverse
                 normaliseN=none|area|unit|maximum|height
                 barformN=open|filled|semi_filled|steps|semi_steps|spikes
                 thickN=<pixels> dashN=dot|dash|...|<a,b,...> xN=<num-expr>
                 weightN=<num-expr> inN=<table> ifmtN=<in-format>
                 istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane layer1=histogram in1=rrlyrae.fits x1=p1
```

`barformN = open|filled|semi_filled|steps|semi_steps|spikes` ***(Form)***

How histogram bars are represented. Note that options using transparent colours may not render very faithfully to some vector formats like PDF and EPS.

The available options are:

- `open`

- `filled`
- `semi_filled`
- `steps`
- `semi_steps`
- `spikes`

[Default: `semi_filled`]

`binsizeN = +<width>|-<count>` ***(BinSizer)***
Configures the width of histogram bins. If the supplied string is a positive number, it is interpreted as a fixed width in the data coordinates of the X axis (if the X axis is logarithmic, the value is a fixed factor). If it is a negative number, then it will be interpreted as the approximate number of bins to display across the width of the plot (though an attempt is made to use only round numbers for bin widths).

When setting this value graphically, you can use either the slider to adjust the bin count or the numeric entry field to fix the bin width.

[Default: `-30`]

`colorN = <rrggbb>|red|blue|...` ***(Color)***
The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`combineN = sum|sum-per-unit|count|...` ***(Combiner)***
Defines how values contributing to the same bin are combined together to produce the value assigned to that bin, and hence its height. The combined values are those given by the `weight` coordinate, but if no weight is supplied, a weighting of unity is assumed.

The available options are:

- `sum`: the sum of all the combined values per bin
- `sum-per-unit`: the sum of all the combined values per unit of bin size
- `count`: the number of non-blank values per bin (weight is ignored)
- `count-per-unit`: the number of non-blank values per unit of bin size (weight is ignored)
- `mean`: the mean of the combined values
- `median`: the median
- `q1`: first quartile
- `q3`: third quartile
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `stdev`: the sample standard deviation of the combined values
- `hit`: 1 if any values present, NaN otherwise (weight is ignored)

[Default: `sum`]

`cumulativeN = none|forward|reverse` ***(Cumulation)***
If set to `forward`/`reverse` the histogram bars plotted are calculated cumulatively; each bin includes the counts from all previous bins working up/down the independent axis.

Note that setting cumulative plotting may not make much sense with some other parameter

values, for instance averaging aggregation modes.

For reasons of backward compatibility, the values `true` and `false` may be used as aliases for `forward` and `none`.

The available options are:

- `none`: The value plotted for each bin uses the samples accumulated in that bin.
- `forward`: The value plotted for each bin uses the samples accumulated all the way from negative infinity to that bin.
- `reverse`: The value plotted for each bin uses the samples accumulated all the way from positive infinity to that bin.

[Default: `none`]

`dashN = dot|dash|...|<a,b,...>` ***(float[])***
Determines the dash pattern of the line drawn. If null (the default), the line is solid.

Possible values for dashed lines are `dot`, `dash`, `longdash`, `dotdash`. You can alternatively supply a comma-separated list of on/off length values such as "`4,2,8,2`".

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is

necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`normaliseN = none|area|unit|maximum|height` ***(Normalisation)***
Defines how, if at all, the bars of histogram-like plots are normalised or otherwise scaled vertically.

Note that some of the normalisation options may not make much sense with some other parameter values, for instance averaging aggregation modes.

The available options are:

- `none`: No normalisation is performed.
- `area`: The total area of histogram bars is normalised to unity. For cumulative plots, this behaves like `height`.
- `unit`: Histogram bars are scaled by the inverse of the bin width in data units. For cumulative plots, this behaves like `none`.
- `maximum`: The height of the tallest histogram bar is normalised to unity. For cumulative plots, this behaves like `height`.
- `height`: The total height of histogram bars is normalised to unity.

[Default: `none`]

`phaseN = <number>` ***(Double)***
Controls where the horizontal zero point for binning is set. For instance if your bin size is 1, this value controls whether bin boundaries are at 0, 1, 2, .. or 0.5, 1.5, 2.5, ... etc.

A value of 0 (or any integer) will result in a bin boundary at X=0 (linear X axis) or X=1 (logarithmic X axis). A fractional value will give a bin boundary at that value multiplied by the bin width.

[Default: `0`]

`thickN = <pixels>` ***(Integer)***
Thickness of plotted line in pixels.

[Default: `2`]

`transparencyN = 0..1` ***(Double)***
Transparency with which components are plotted, in the range 0 (opaque) to 1 (invisible). The value is 1-alpha.

[Default: `0`]

`weightN = <num-expr>` ***(String)***
Weighting of data points. If supplied, each point contributes a value to the histogram equal to the data value multiplied by this coordinate. If not supplied, the effect is the same as supplying a fixed value of one.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xN = <num-expr>` ***(String)***
Horizontal coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.31 kde

Plots a Discrete Kernel Density Estimate giving a smoothed frequency of data values along the horizontal axis, using a fixed-width smoothing kernel. This is a generalisation of a histogram in which the bins are always 1 pixel wide, and a smoothing kernel is applied to each bin. The width and shape of the kernel may be varied.

This is suitable for cases where the division into discrete bins done by a normal histogram is unnecessary or troublesome.

Note this is not a true Kernel Density Estimate, since, for performance reasons, the smoothing is applied to the (pixel-width) bins rather than to each data sample. The deviation from a true KDE caused by this quantisation will be at the pixel level, hence in most cases not visually apparent.

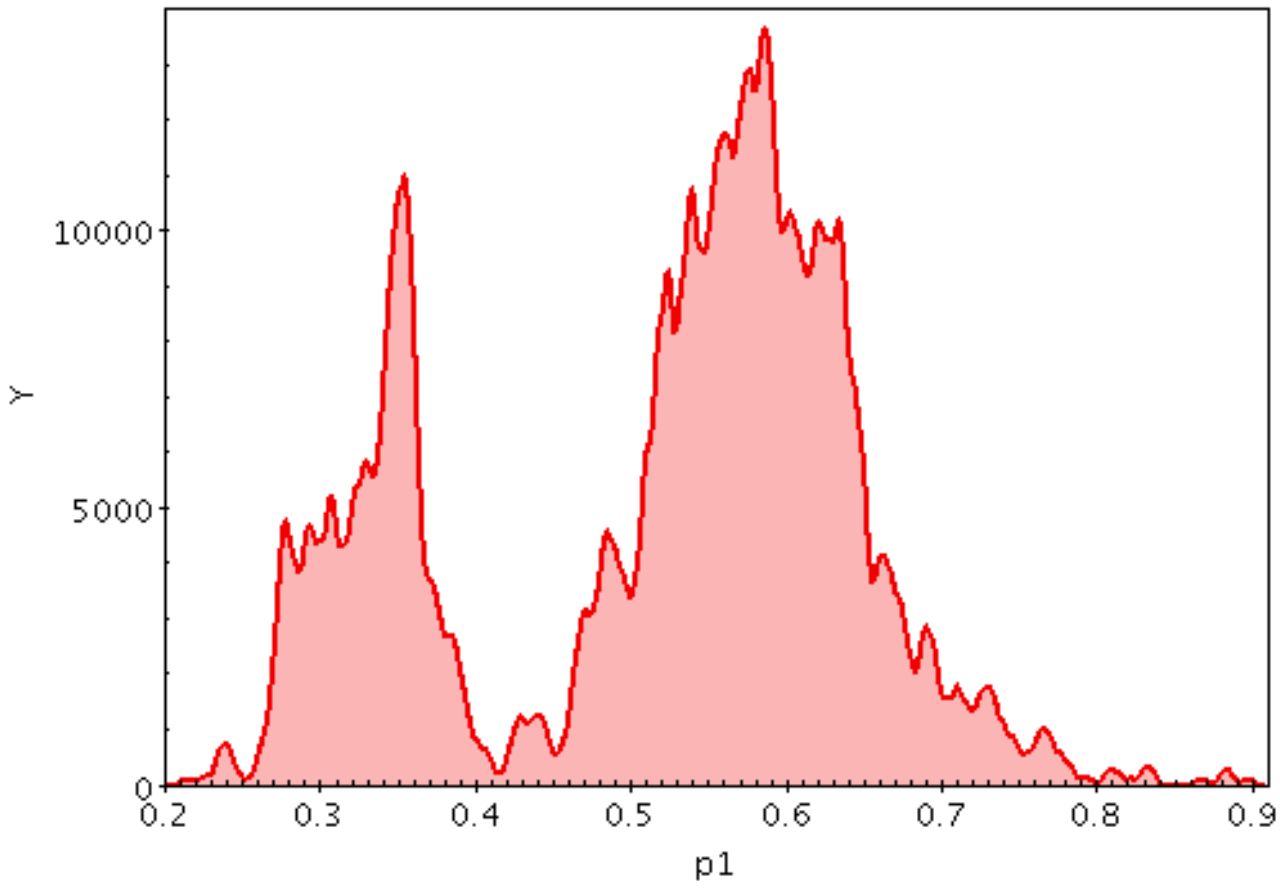
A weighting may be applied to the calculated levels by supplying the `weight` coordinate. In this case you can choose how these weights are aggregated in each pixel bin using the `combine` parameter. The result is something like a smoothed version of the corresponding weighted histogram. Note that some combinations of the available parameters (e.g. a normalised cumulative median-aggregated KDE) may not make much visual sense.

Usage Overview:

```
layerN=kde colorN=<rrggbb>|red|blue|... transparencyN=0..1
           smoothN=+<width>|-<count> combineN=sum|sum-per-unit|count|...
           kernelN=square|linear|epanechnikov|cos|cos2|gauss3|gauss6
           cumulativeN=none|forward|reverse
           normaliseN=none|area|unit|maximum|height fillN=solid|line|semi
           thickN=<pixels> xN=<num-expr> weightN=<num-expr> inN=<table>
           ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane ymin=0 layer1=kde in1=rrlyrae.fits x1=p1
```

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`combineN = sum|sum-per-unit|count|...` ***(Combiner)***

Defines how values contributing to the same bin are combined together to produce the value assigned to that bin, and hence its height. The bins in this case are 1-pixel wide, so lack much physical significance. This means that while some combination modes, such as `sum-per-unit` and `mean` make sense, others such as `sum` do not.

The combined values are those given by the `weight` coordinate, but if no weight is supplied, a weighting of unity is assumed.

The available options are:

- `sum`: the sum of all the combined values per bin
- `sum-per-unit`: the sum of all the combined values per unit of bin size
- `count`: the number of non-blank values per bin (weight is ignored)
- `count-per-unit`: the number of non-blank values per unit of bin size (weight is ignored)

- `mean`: the mean of the combined values
- `median`: the median
- `q1`: first quartile
- `q3`: third quartile
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `stdev`: the sample standard deviation of the combined values
- `hit`: 1 if any values present, NaN otherwise (weight is ignored)

[Default: `sum-per-unit`]

`cumulativeN = none|forward|reverse` ***(Cumulation)***

If set to `forward`/`reverse` the histogram bars plotted are calculated cumulatively; each bin includes the counts from all previous bins working up/down the independent axis.

Note that setting cumulative plotting may not make much sense with some other parameter values, for instance averaging aggregation modes.

For reasons of backward compatibility, the values `true` and `false` may be used as aliases for `forward` and `none`.

The available options are:

- `none`: The value plotted for each bin uses the samples accumulated in that bin.
- `forward`: The value plotted for each bin uses the samples accumulated all the way from negative infinity to that bin.
- `reverse`: The value plotted for each bin uses the samples accumulated all the way from positive infinity to that bin.

[Default: `none`]

`fillN = solid|line|semi` ***(FillMode)***

How the density function is represented.

The available options are:

- `solid`: area between level and axis is filled with solid colour
- `line`: level is marked by a wiggly line
- `semi`: level is marked by a wiggly line, and area below it is filled with a transparent colour

[Default: `semi`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for

scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`kernelN = square|linear|epanechnikov|cos|cos2|gauss3|gauss6` ***(Kernel1dShape)***

The functional form of the smoothing kernel. The functions listed refer to the unscaled shape; all kernels are normalised to give a total area of unity.

The available options are:

- `square`: Uniform value: f(x)=1, |x|=0..1
- `linear`: Triangle: f(x)=1-|x|, |x|=0..1
- `epanechnikov`: Parabola: f(x)=1-x*x, |x|=0..1
- `cos`: Cosine: f(x)=cos(x*pi/2), |x|=0..1
- `cos2`: Cosine squared: f(x)=cos^2(x*pi/2), |x|=0..1
- `gauss3`: Gaussian truncated at 3.0 sigma: f(x)=exp(-x*x/2), |x|=0..3
- `gauss6`: Gaussian truncated at 6.0 sigma: f(x)=exp(-x*x/2), |x|=0..6

[Default: `epanechnikov`]

`normaliseN = none|area|unit|maximum|height` ***(Normalisation)***

Defines how, if at all, the bars of histogram-like plots are normalised or otherwise scaled vertically.

Note that some of the normalisation options may not make much sense with some other parameter values, for instance averaging aggregation modes.

The available options are:

- `none`: No normalisation is performed.
- `area`: The total area of histogram bars is normalised to unity. For cumulative plots, this behaves like `height`.
- `unit`: Histogram bars are scaled by the inverse of the bin width in data units. For cumulative plots, this behaves like `none`.
- `maximum`: The height of the tallest histogram bar is normalised to unity. For cumulative

plots, this behaves like `height`.
- `height`: The total height of histogram bars is normalised to unity.

[Default: `none`]

`smoothN = +<width>|-<count>` ***(BinSizer)***
Configures the smoothing width for kernel density estimation. This is the characteristic width of the kernel function to be convolved with the density to produce the visible plot.

If the supplied value is a positive number it is interpreted as a fixed width in the data coordinates of the X axis (if the X axis is logarithmic, the value is a fixed factor). If it is a negative number, then it will be interpreted as the approximate number of smooothing widths that fit in the width of the visible plot (i.e. plot width / smoothing width). If the value is zero, no smoothing is applied.

When setting this value graphically, you can use either the slider to adjust the bin count or the numeric entry field to fix the bin width.

[Default: `-100`]

`thickN = <pixels>` ***(Integer)***
Thickness of plotted line in pixels.

[Default: `2`]

`transparencyN = 0..1` ***(Double)***
Transparency with which components are plotted, in the range 0 (opaque) to 1 (invisible). The value is 1-alpha.

[Default: `0`]

`weightN = <num-expr>` ***(String)***
Weighting of data points. If supplied, each point contributes a value to the histogram equal to the data value multiplied by this coordinate. If not supplied, the effect is the same as supplying a fixed value of one.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xN = <num-expr>` ***(String)***
Horizontal coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.32 `knn`

Plots a Discrete Kernel Density Estimate giving a smoothed frequency of data values along the horizontal axis, using an adaptive (K-Nearest-Neighbours) smoothing kernel. This is a generalisation of a histogram in which the bins are always 1 pixel wide, and a smoothing kernel is applied to each bin. The width and shape of the kernel may be varied.

The K-Nearest-Neighbour figure gives the number of points in each direction to determine the width of the smoothing kernel for smoothing each bin. Upper and lower limits for the kernel width are also supplied; if the upper and lower limits are equal, this is equivalent to a fixed-width kernel.

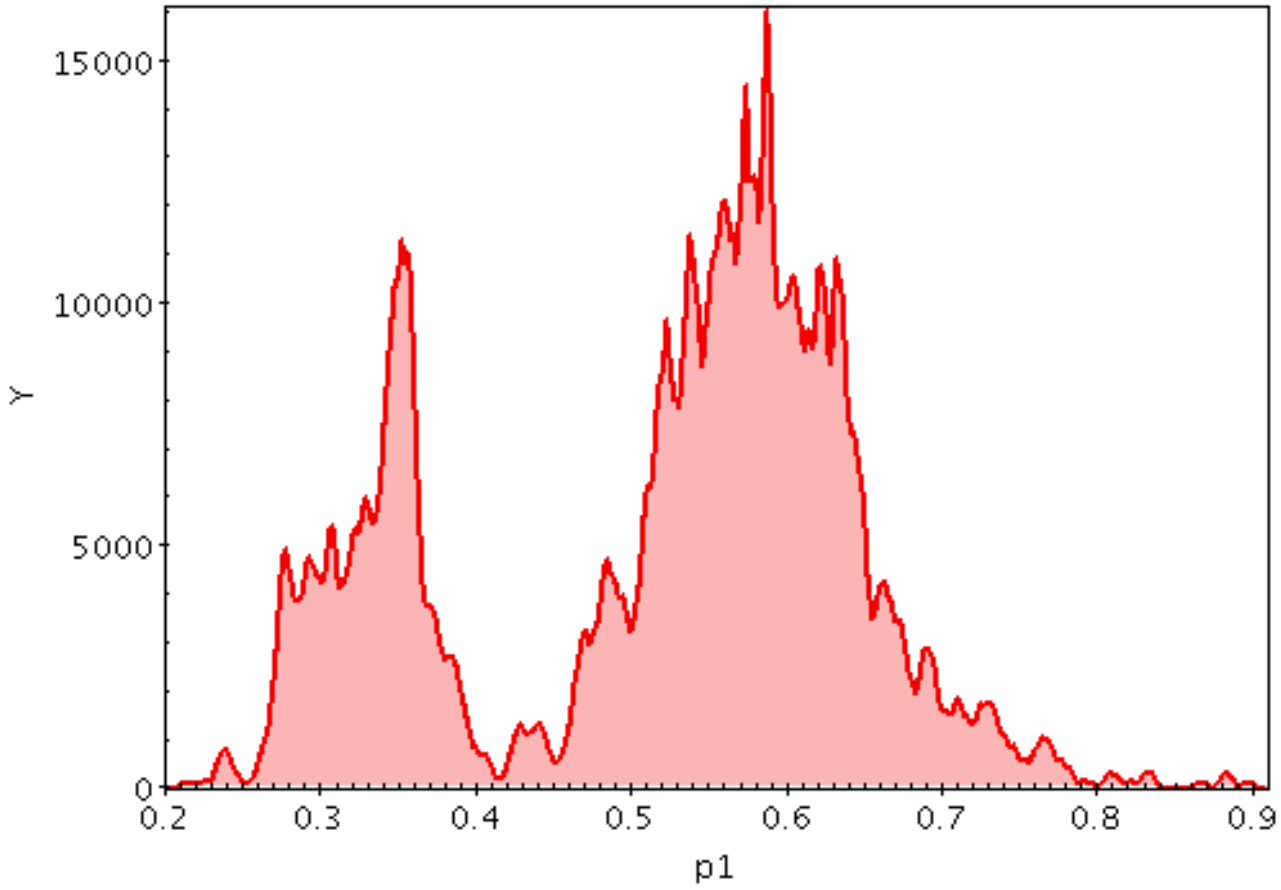
Note this is not a true Kernel Density Estimate, since, for performance reasons, the smoothing is applied to the (pixel-width) bins rather than to each data sample. The deviation from a true KDE caused by this quantisation will be at the pixel level, hence in most cases not visually apparent.

Usage Overview:

```
layerN=knn colorN=<rrggbb>|red|blue|... transparencyN=0..1 knnN=<number>
           symmetricN=true|false minsmoothN=+<width>|-<count>
           maxsmoothN=+<width>|-<count>
           kernelN=square|linear|epanechnikov|cos|cos2|gauss3|gauss6
           cumulativeN=none|forward|reverse
           normaliseN=none|area|unit|maximum|height fillN=solid|line|semi
           thickN=<pixels> xN=<num-expr> weightN=<num-expr> inN=<table>
           ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane layer1=knn in1=rrlyrae.fits x1=p1
```

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`cumulativeN = none|forward|reverse` ***(Cumulation)***

If set to `forward`/`reverse` the histogram bars plotted are calculated cumulatively; each bin

includes the counts from all previous bins working up/down the independent axis.

Note that setting cumulative plotting may not make much sense with some other parameter values, for instance averaging aggregation modes.

For reasons of backward compatibility, the values `true` and `false` may be used as aliases for `forward` and `none`.

The available options are:

- `none`: The value plotted for each bin uses the samples accumulated in that bin.
- `forward`: The value plotted for each bin uses the samples accumulated all the way from negative infinity to that bin.
- `reverse`: The value plotted for each bin uses the samples accumulated all the way from positive infinity to that bin.

[Default: `none`]

`fillN = solid|line|semi` ***(FillMode)***
How the density function is represented.

The available options are:

- `solid`: area between level and axis is filled with solid colour
- `line`: level is marked by a wiggly line
- `semi`: level is marked by a wiggly line, and area below it is filled with a transparent colour

[Default: `semi`]

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the

end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`kernelN = square|linear|epanechnikov|cos|cos2|gauss3|gauss6` ***(Kernel1dShape)***

The functional form of the smoothing kernel. The functions listed refer to the unscaled shape; all kernels are normalised to give a total area of unity.

The available options are:

- `square`: Uniform value: f(x)=1, |x|=0..1
- `linear`: Triangle: f(x)=1-|x|, |x|=0..1
- `epanechnikov`: Parabola: f(x)=1-x*x, |x|=0..1
- `cos`: Cosine: f(x)=cos(x*pi/2), |x|=0..1
- `cos2`: Cosine squared: f(x)=cos^2(x*pi/2), |x|=0..1
- `gauss3`: Gaussian truncated at 3.0 sigma: f(x)=exp(-x*x/2), |x|=0..3
- `gauss6`: Gaussian truncated at 6.0 sigma: f(x)=exp(-x*x/2), |x|=0..6

[Default: `epanechnikov`]

`knnN = <number>` ***(Double)***

Sets the number of nearest neighbours to count away from a sample point to determine the width of the smoothing kernel at that point. For the symmetric case this is the number of nearest neighbours summed over both directions, and for the asymmetric case it is the number in a single direction.

The threshold is actually the weighted total of samples; for unweighted (`weight=1`) bins that is equivalent to the number of samples.

[Default: `100`]

`maxsmoothN = +<width>|-<count>` ***(BinSizer)***

Fixes the maximum size of the smoothing kernel. This functions as an upper limit on the distance that is otherwise determined by searching for the K nearest neighbours at each sample point.

If the supplied value is a positive number it is interpreted as a fixed width in the data coordinates of the X axis (if the X axis is logarithmic, the value is a fixed factor). If it is a negative number, then it will be interpreted as the approximate number of smooothing widths that fit in the width of the visible plot (i.e. plot width / smoothing width). If the value is zero, no smoothing is applied.

When setting this value graphically, you can use either the slider to adjust the bin count or the numeric entry field to fix the bin width.

[Default: `-100`]

`minsmoothN = +<width>|-<count>` ***(BinSizer)***

Fixes the minimum size of the smoothing kernel. This functions as a lower limit on the distance that is otherwise determined by searching for the K nearest neighbours at each sample

point.

If the supplied value is a positive number it is interpreted as a fixed width in the data coordinates of the X axis (if the X axis is logarithmic, the value is a fixed factor). If it is a negative number, then it will be interpreted as the approximate number of smooothing widths that fit in the width of the visible plot (i.e. plot width / smoothing width). If the value is zero, no smoothing is applied.

When setting this value graphically, you can use either the slider to adjust the bin count or the numeric entry field to fix the bin width.

[Default: `0`]

`normaliseN = none|area|unit|maximum|height` ***(Normalisation)***
Defines how, if at all, the bars of histogram-like plots are normalised or otherwise scaled vertically.

Note that some of the normalisation options may not make much sense with some other parameter values, for instance averaging aggregation modes.

The available options are:

- `none`: No normalisation is performed.
- `area`: The total area of histogram bars is normalised to unity. For cumulative plots, this behaves like `height`.
- `unit`: Histogram bars are scaled by the inverse of the bin width in data units. For cumulative plots, this behaves like `none`.
- `maximum`: The height of the tallest histogram bar is normalised to unity. For cumulative plots, this behaves like `height`.
- `height`: The total height of histogram bars is normalised to unity.

[Default: `none`]

`symmetricN = true|false` ***(Boolean)***
If true, the nearest neigbour search is carried out in both directions, and the kernel is symmetric. If false, the nearest neigbour search is carried out separately in the positive and negative directions, and the kernel width is accordingly different in the positive and negative directions.

[Default: `true`]

`thickN = <pixels>` ***(Integer)***
Thickness of plotted line in pixels.

[Default: `2`]

`transparencyN = 0..1` ***(Double)***
Transparency with which components are plotted, in the range 0 (opaque) to 1 (invisible). The value is 1-alpha.

[Default: `0`]

`weightN = <num-expr>` ***(String)***
Weighting of data points. If supplied, each point contributes a value to the histogram equal to the data value multiplied by this coordinate. If not supplied, the effect is the same as supplying a fixed value of one.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xN = <num-expr>` ***(String)***
Horizontal coordinate.

The value is a numeric algebraic expression based on column names as described in Section

10.

8.3.33 `densogram`

Represents smoothed density of data values along the horizontal axis using a colourmap. This is like a Kernel Density Estimate (smoothed histogram with bins 1 pixel wide), but instead of representing the data extent vertically as bars or a line, values are represented by a fixed-size pixel-width column of a colour from a colour map. A smoothing kernel, whose width and shape may be varied, is applied to each data point.

A weighting may be applied to the calculated levels by supplying the `weight` coordinate. In this case you can choose how these weights are aggregated in each pixel bin using the `combine` parameter. The result is something like a smoothed version of the corresponding weighted histogram. Note that some combinations of the available parameters (e.g. a normalised cumulative median-aggregated KDE) may not make much visual sense.

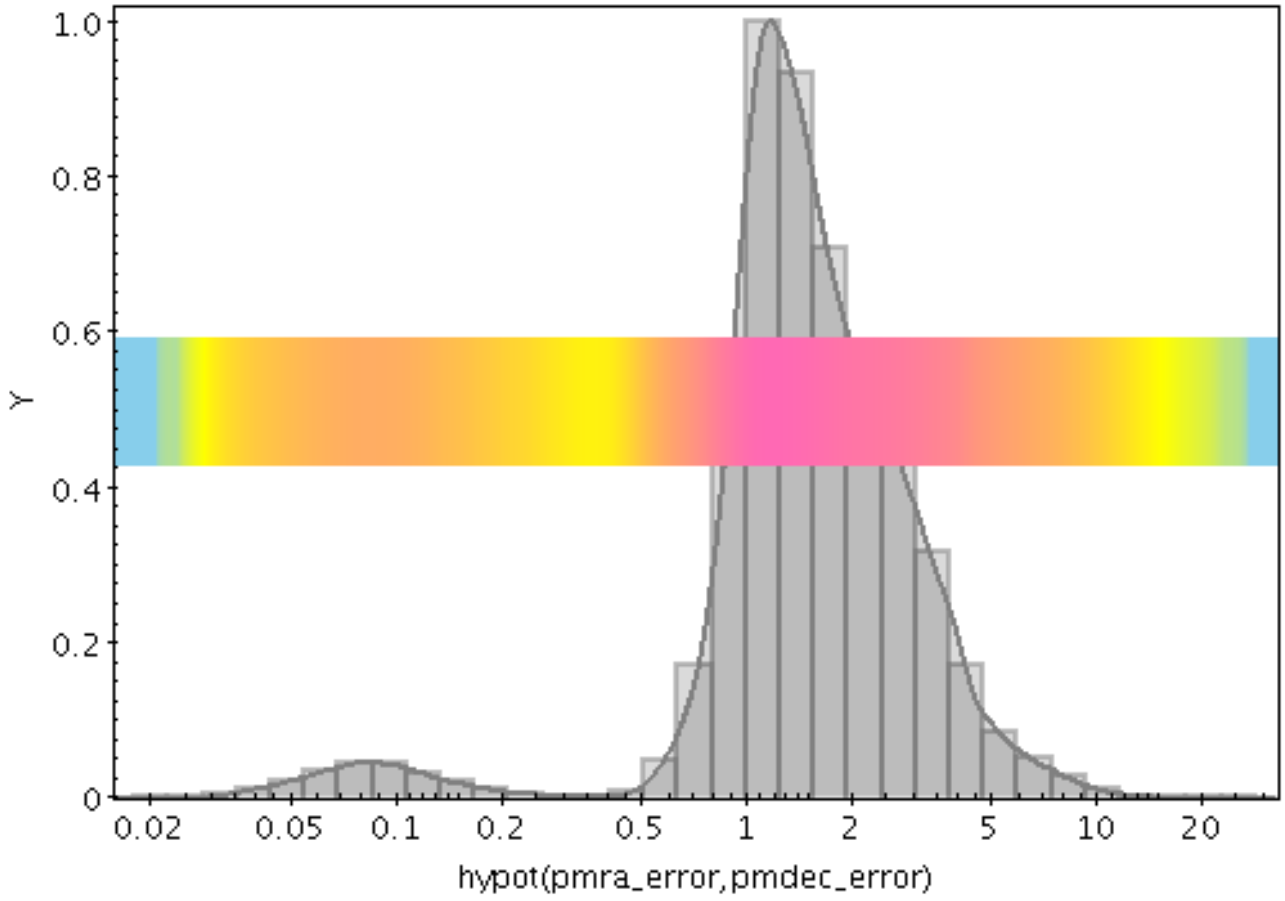
This is a rather unconventional way to represent density data, and this plotting mode is probably not very useful. But hey, nobody's forcing you to use it.

Usage Overview:

```
layerN=densogram colorN=<rrggbb>|red|blue|... smoothN=+<width>|-<count>
                 kernelN=square|linear|epanechnikov|cos|cos2|gauss3|gauss6
                 densemapN=<map-name>|<color>-<color>[-<color>...]
                 denseclipN=<lo>,<hi> denseflipN=true|false
                 densequantN=<number> densesubN=<lo>,<hi>
                 densefuncN=log|linear|histogram|histolog|sqrt|square|acos|cos
                 cumulativeN=none|forward|reverse sizeN=<pixels>
                 posN=<fraction> xN=<num-expr> weightN=<num-expr>
                 inN=<table> ifmtN=<in-format> istreamN=true|false
                 icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in=tgas_source.fits x=hypot(pmra_error,pmdec_error)
                  xlog=true normalise=maximum
                  color=grey layer1=histogram layer2=kde
                  layer3=densogram densemap3=skyblue-yellow-hotpink densefunc3=log
                  size3=50 pos3=0.5
```

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`cumulativeN = none|forward|reverse` ***(Cumulation)***

If set to `forward`/`reverse` the histogram bars plotted are calculated cumulatively; each bin includes the counts from all previous bins working up/down the independent axis.

Note that setting cumulative plotting may not make much sense with some other parameter values, for instance averaging aggregation modes.

For reasons of backward compatibility, the values `true` and `false` may be used as aliases for `forward` and `none`.

The available options are:

- `none`: The value plotted for each bin uses the samples accumulated in that bin.
- `forward`: The value plotted for each bin uses the samples accumulated all the way from negative infinity to that bin.
- `reverse`: The value plotted for each bin uses the samples accumulated all the way from positive infinity to that bin.

[Default: `none`]

`denseclipN = <lo>,<hi>` ***(Subrange)***
Defines a subrange of the colour ramp to be used for Density shading. The value is specified as a (low,high) comma-separated pair of two numbers between 0 and 1.

If the full range `0,1` is used, the whole range of colours specified by the selected shader will be used. But if for instance a value of `0,0.5` is given, only those colours at the left hand end of the ramp will be seen.

If the null (default) value is chosen, a default clip will be used. This generally covers most or all of the range 0-1 but for colour maps which fade to white, a small proportion of the lower end may be excluded, to ensure that all the colours are visually distinguishable from a white background. This default is usually a good idea if the colour map is being used with something like a scatter plot, where markers are plotted against a white background. However, for something like a density map when the whole plotting area is tiled with colours from the map, it may be better to supply the whole range `0,1` explicitly.

`denseflipN = true|false` ***(Boolean)***
If true, the colour map on the Density axis will be reversed.

[Default: `false`]

`densefuncN = log|linear|histogram|histolog|sqrt|square|acos|cos` ***(Scaling)***
Defines the way that values in the Density range are mapped to the selected colour ramp.

The available options are:

- `log`: Logarithmic scaling
- `linear`: Linear scaling
- `histogram`: Scaling follows data distribution, with linear axis
- `histolog`: Scaling follows data distribution, with logarithmic axis
- `sqrt`: Square root scaling
- `square`: Square scaling
- `acos`: Arccos Scaling
- `cos`: Cos Scaling

For all these options, the full range of data values is used, and displayed on the colour bar if applicable (though it can be restricted using the `densesub` option) The `Linear`, `Log`, `Square` and `Sqrt` options just apply the named function to the full data range. The histogram options on the other hand use a scaling function that corresponds to the actual distribution of the data, so that there are about the same number of points (or pixels, or whatever is being scaled) of each colour. The histogram options are somewhat more expensive, but can be a good choice if you are exploring data whose distribution is unknown or not well-behaved over its min-max range. The `Histogram` and `HistoLog` options both assign the colours in the same way, but they display the colour ramp with linear or logarithmic annotation respectively; the `HistoLog` option also ignores non-positive values.

[Default: `linear`]

`densemapN = <map-name>|<color>-<color>[-<color>...]` ***(Shader)***
Color map used for Density axis shading.

A mixed bag of colour ramps are available: `inferno`, `magma`, `plasma`, `viridis`, `cividis`, `cubehelix`, `sron`, `rainbow`, `rainbow2`, `rainbow3`, `pastel`, `accent`, `gnuplot`, `gnuplot2`,

, `set1`, `paired`, `hotcold`, `rdbu`, `piyg`, `brbg`, `cyan-magenta`, `red-blue`, `brg`, `heat`, `cold`, `light`, `greyscale`, `colour`, `standard`, `bugn`, `bupu`, `orrd`, `pubu`, `purd`, `painbow`, `huecl`, `hue`, `intensity`, `rgb_red`, `rgb_green`, `rgb_blue`, `hsv_h`, `hsv_s`, `hsv_v`, `yuv_y`, `yuv_u`, `yuv_v`, `scale_hsv_s`, `scale_hsv_v`, `scale_yuv_y`, `mask`, `blacker`, `whiter`, `transparency`. *Note:* many of these, including rainbow-like ones, are frowned upon by the visualisation community.

You can also construct your own custom colour map by giving a sequence of colour names separated by minus sign ("`-`") characters. In this case the ramp is a linear interpolation between each pair of colours named, using the same syntax as when specifying a colour value. So for instance "`yellow-hotpink-#0000ff`" would shade from yellow via hot pink to blue.

[Default: `inferno`]

`densequantN = <number>` ***(Double)***

Allows the colour map used for the Density axis to be quantised. If an integer value N is chosen then the colour map will be viewed as N discrete evenly-spaced levels, so that only N different colours will appear in the plot. This can be used to generate a contour-like effect, and may make it easier to trace the boundaries of regions of interest by eye.

If left blank, the colour map is nominally continuous (though in practice it may be quantised to a medium-sized number like 256).

`densesubN = <lo>,<hi>` ***(Subrange)***

Defines a normalised adjustment to the data range of the Density axis. The value may be specified as a comma-separated pair of two numbers, giving the lower and upper bounds of the range of of interest respectively. This sub-range is applied to the data range that would otherwise be used, either automatically calculated or explicitly supplied; zero corresponds to the lower bound and one to the upper.

The default value "`0,1`" therefore has no effect. The range could be restricted to its lower half with the value `0,0.5`.

[Default: `0,1`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.

- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`kernelN = square|linear|epanechnikov|cos|cos2|gauss3|gauss6` ***(Kernel1dShape)***

The functional form of the smoothing kernel. The functions listed refer to the unscaled shape; all kernels are normalised to give a total area of unity.

The available options are:

- `square`: Uniform value: f(x)=1, |x|=0..1
- `linear`: Triangle: f(x)=1-|x|, |x|=0..1
- `epanechnikov`: Parabola: f(x)=1-x*x, |x|=0..1
- `cos`: Cosine: f(x)=cos(x*pi/2), |x|=0..1
- `cos2`: Cosine squared: f(x)=cos^2(x*pi/2), |x|=0..1
- `gauss3`: Gaussian truncated at 3.0 sigma: f(x)=exp(-x*x/2), |x|=0..3
- `gauss6`: Gaussian truncated at 6.0 sigma: f(x)=exp(-x*x/2), |x|=0..6

[Default: `epanechnikov`]

`posN = <fraction>` ***(Double)***

Determines where on the plot region the density bar appears. The value should be in the range 0..1; zero corresponds to the bottom of the plot and one to the top.

[Default: `0.05`]

`sizeN = <pixels>` ***(Integer)***

Height of the density bar in pixels.

[Default: `12`]

`smoothN = +<width>|-<count>` ***(BinSizer)***

Configures the smoothing width for kernel density estimation. This is the characteristic width of the kernel function to be convolved with the density to produce the visible plot.

If the supplied value is a positive number it is interpreted as a fixed width in the data coordinates of the X axis (if the X axis is logarithmic, the value is a fixed factor). If it is a negative number, then it will be interpreted as the approximate number of smooothing widths that fit in the width of the visible plot (i.e. plot width / smoothing width). If the value is zero, no smoothing is applied.

When setting this value graphically, you can use either the slider to adjust the bin count or the numeric entry field to fix the bin width.

[Default: `-100`]

`weightN = <num-expr>` ***(String)***
Weighting of data points. If supplied, each point contributes a value to the histogram equal to the data value multiplied by this coordinate. If not supplied, the effect is the same as supplying a fixed value of one.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xN = <num-expr>` ***(String)***
Horizontal coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.34 `gaussian`

Plots a best fit Gaussian to the histogram of a sample of data. In fact, all this plotter does is to calculate the mean and standard deviation of the sample, and plot the corresponding Gaussian curve. The mean and standard deviation values are reported by the plot.

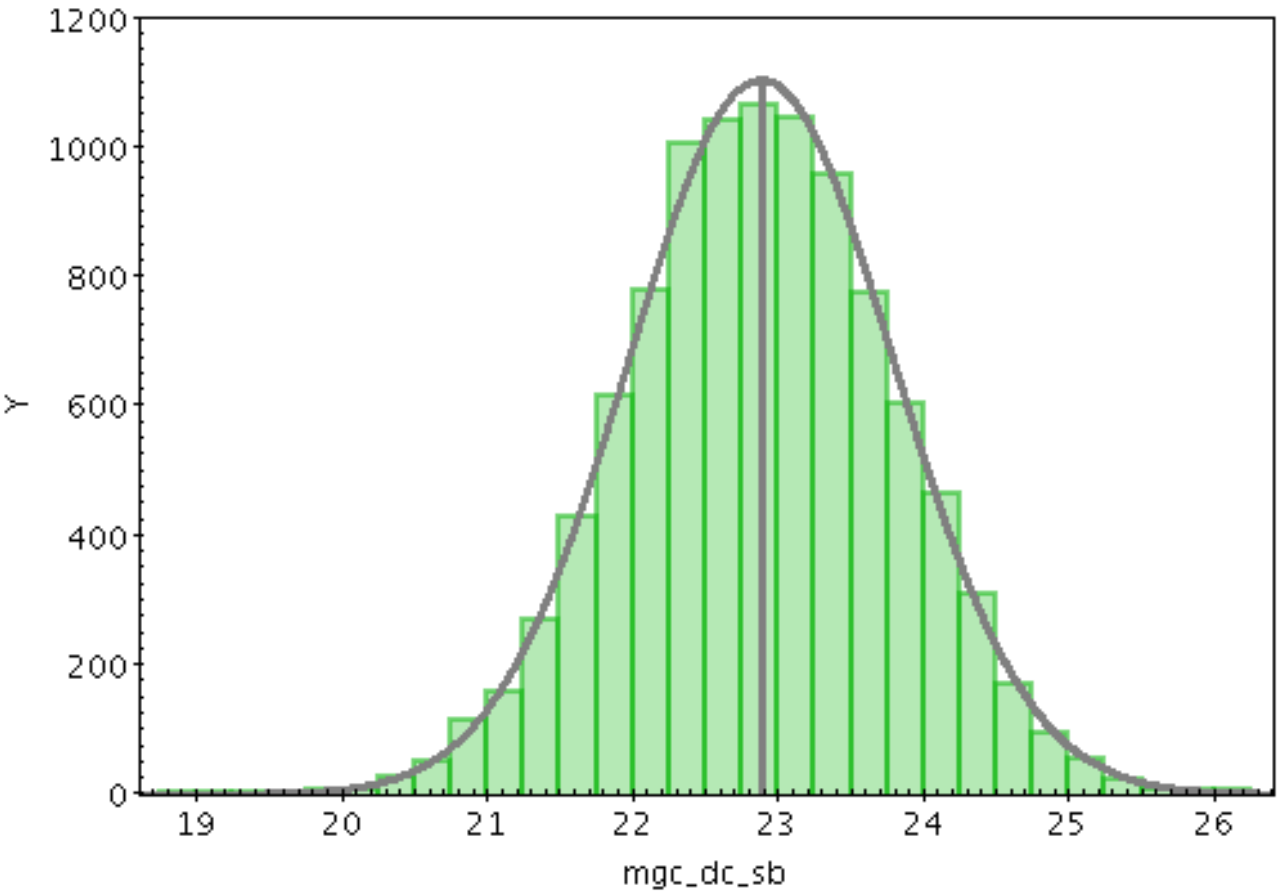
The `normalise` config option, perhaps in conjunction with `binsize`, can be used to scale the height of the plotted curve in data units. In this case, `binsize` just describes the bar width of a notional histogram whose outline the plotted Gaussian should try to fit, and is only relevant for some of the normalisation options.

Usage Overview:

```
layerN=gaussian colorN=<rrggbb>|red|blue|... showmeanN=true|false
                thickN=<pixels> dashN=dot|dash|...|<a,b,...>
                antialiasN=true|false
                normaliseN=none|area|unit|maximum|height
                binsizeN=+<width>|-<count> xN=<num-expr> weightN=<num-expr>
                inN=<table> ifmtN=<in-format> istreamN=true|false
                icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane in=mgc_ok.fits x=mgc_dc_sb
                  layer1=histogram color1=green
                  layer2=gaussian color2=grey thick2=3
                  ymax=1200
```

`antialiasN = true|false` ***(Boolean)***

If true, plotted lines are drawn with antialising. Antialised lines look smoother, but may take perceptibly longer to draw. Only has any effect for bitmapped output formats.

[Default: `false`]

`binsizeN = +<width>|-<count>` ***(BinSizer)***

Configures the width of histogram bins. If the supplied string is a positive number, it is interpreted as a fixed width in the data coordinates of the X axis (if the X axis is logarithmic, the value is a fixed factor). If it is a negative number, then it will be interpreted as the approximate number of bins to display across the width of the plot (though an attempt is made to use only round numbers for bin widths).

When setting this value graphically, you can use either the slider to adjust the bin count or the numeric entry field to fix the bin width.

[Default: `-30`]

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed

by "#" or "0x", giving red, green and blue intensities, e.g. "ff00ff", "#ff00ff" or "0xff00ff" for magenta.

[Default: red]

dashN = dot|dash|...|<a,b,...> ***(float[])***
Determines the dash pattern of the line drawn. If null (the default), the line is solid.

Possible values for dashed lines are dot, dash, longdash, dotdash. You can alternatively supply a comma-separated list of on/off length values such as "4,2,8,2".

icmdN = <cmds> ***(ProcessingStep[])***
Specifies processing to be performed on the layer N input table as specified by parameter inN. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ifmtN = <in-format> ***(String)***
Specifies the format of the input table as specified by parameter inN. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value (auto) (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: (auto)]

inN = <table> ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "-", meaning standard input. In this case the input format must be given explicitly using the ifmtN parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form :<scheme-name>:<scheme-args>.
- A system command line with either a "<" character at the start, or a "|" character at the end ("<syscmd" or "syscmd|"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***
If set true, the input table specified by the inN parameter will be read as a stream. It is necessary to give the ifmtN parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: false]

normaliseN = none|area|unit|maximum|height ***(Normalisation)***

Defines how, if at all, the bars of histogram-like plots are normalised or otherwise scaled vertically.

Note that some of the normalisation options may not make much sense with some other parameter values, for instance averaging aggregation modes.

The available options are:

- `none`: No normalisation is performed.
- `area`: The total area of histogram bars is normalised to unity. For cumulative plots, this behaves like `height`.
- `unit`: Histogram bars are scaled by the inverse of the bin width in data units. For cumulative plots, this behaves like `none`.
- `maximum`: The height of the tallest histogram bar is normalised to unity. For cumulative plots, this behaves like `height`.
- `height`: The total height of histogram bars is normalised to unity.

[Default: `none`]

`showmeanN = true|false` ***(Boolean)***
If true, a line is drawn at the position of the calculated mean.

[Default: `true`]

`thickN = <pixels>` ***(Integer)***
Thickness of plotted line in pixels.

[Default: `1`]

`weightN = <num-expr>` ***(String)***
Weighting of data points. If supplied, each point contributes a value to the histogram equal to the data value multiplied by this coordinate. If not supplied, the effect is the same as supplying a fixed value of one.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xN = <num-expr>` ***(String)***
Horizontal coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.35 `function`

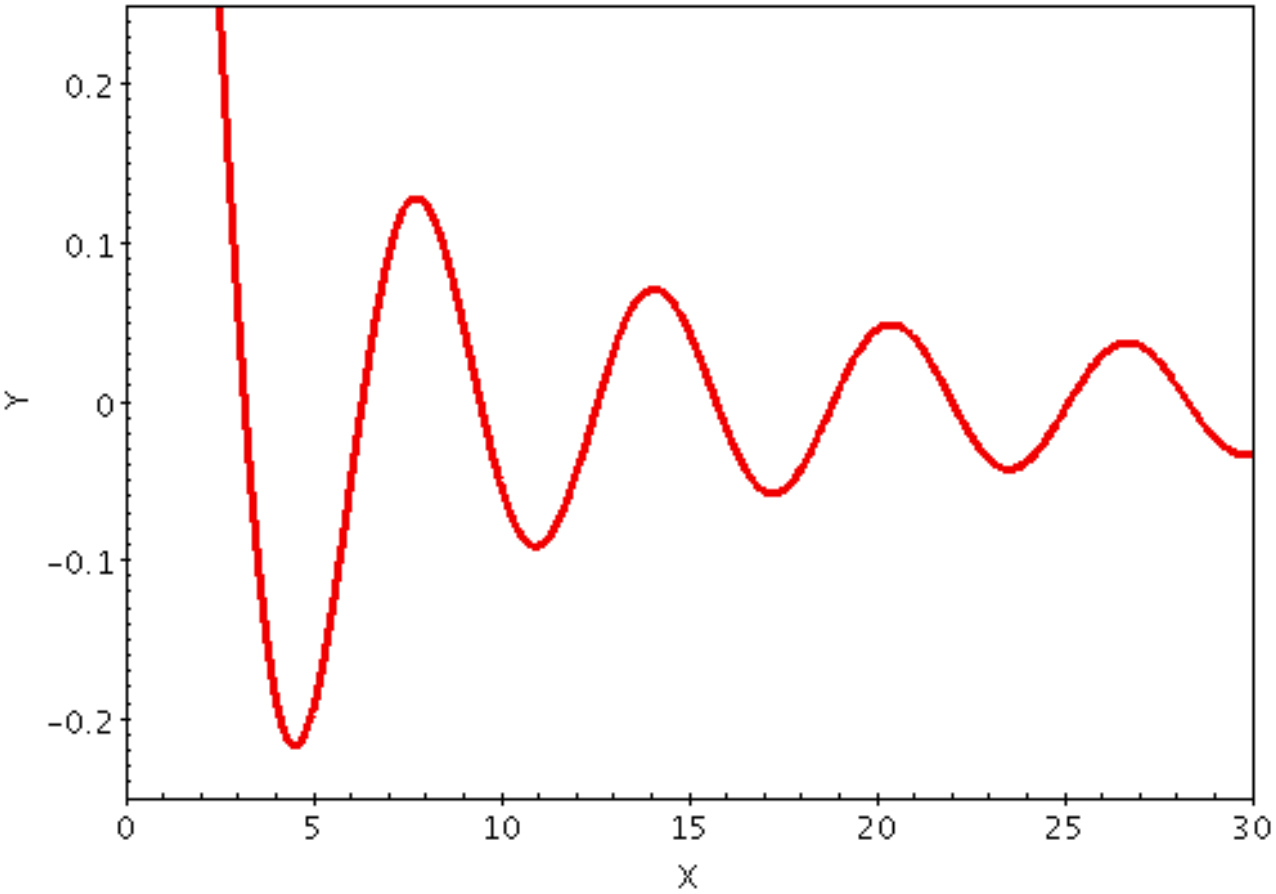
Plots an analytic function. This layer is currently only available for the Plane plots (including histogram).

Usage Overview:

```
layerN=function axisN=Horizontal|Vertical xnameN=<name> fexprN=<expr>
                colorN=<rrggbb>|red|blue|... thickN=<pixels>
                dashN=dot|dash|...|<a,b,...> antialiasN=true|false
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane layer1=function fexpr1=sin(x)/x thick1=3
                  xmin=0 xmax=30 ymin=-0.25 ymax=0.25
```

`antialiasN = true|false` ***(Boolean)***

If true, plotted lines are drawn with antialising. Antialised lines look smoother, but may take perceptibly longer to draw. Only has any effect for bitmapped output formats.

[Default: `false`]

`axisN = Horizontal|Vertical` ***(FuncAxis)***

Which axis the independent variable varies along. Options are currently `Horizontal` and `Vertical`.

[Default: `Horizontal`]

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`dashN = dot|dash|...|<a,b,...>` ***(float[])***

Determines the dash pattern of the line drawn. If null (the default), the line is solid.

Possible values for dashed lines are `dot`, `dash`, `longdash`, `dotdash`. You can alternatively supply a comma-separated list of on/off length values such as "`4,2,8,2`".

`fexprN = <expr>` ***(String)***
An expression using TOPCAT's expression language in terms of the independent variable to define the function. This expression must be standalone - it cannot reference any tables.

`thickN = <pixels>` ***(Integer)***
Thickness of plotted line in pixels.

[Default: `1`]

`xnameN = <name>` ***(String)***
Name of the independent variable for use in the function expression. This is typically `x` for a horizontal independent variable and `y` for a vertical independent variable, but any string that is a legal expression language identifier (starts with a letter, continues with letters, numbers, underscores) can be used.

[Default: `x`]

8.3.36 `skyvector`

Plots directed lines from the data position given delta values for the coordinates The plotted markers are typically little arrows, but there are other options.

The dimensions of the plotted vectors are given by the `dlon` and `dlat` coordinates. The units of these values are specified using the `unit` option. If only the relative rather than the absolute sizes are required on the plot, or if the units are not known, the special value `unit=scaled` may be used; this applies a non-physical scaling factor to make the vectors appear at some reasonable size in the plot. When `unit=scaled` vectors will keep approximately the same screen size during zoom operations; when one of the angular units is chosen, they will keep the same size in data coordinates.

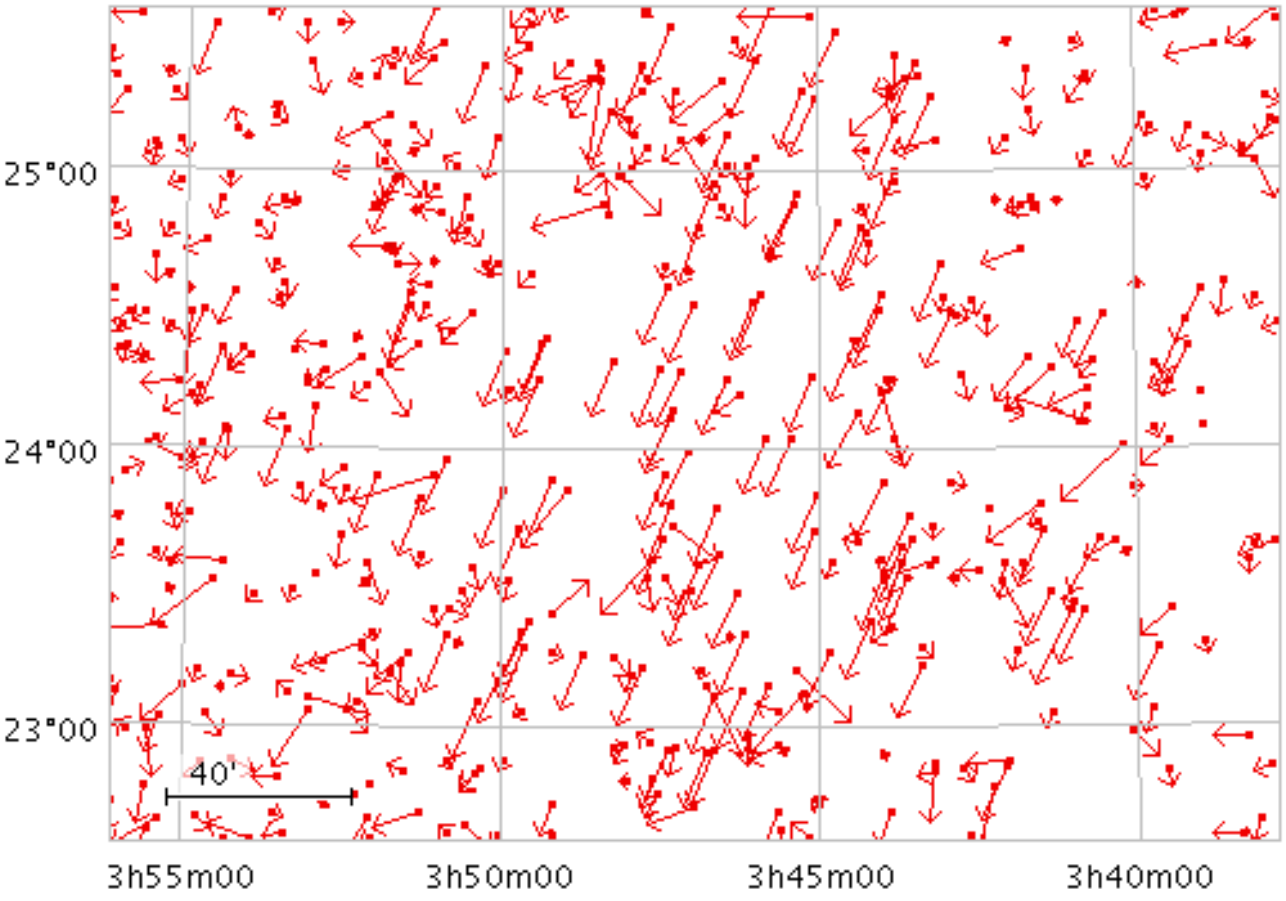
Additionally, the `scale` option may be used to scale all the plotted vectors by a given factor to make them all larger or smaller.

Usage Overview:

```
layerN=skyvector arrowN=small_arrow|medium_arrow|... thickN=<int-value>
                 unitN=scaled|radian|degree|minute|arcsec|mas|uas
                 scaleN=<number>
                 shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-par
                 lonN=<deg-expr> latN=<deg-expr> dlonN=<num-expr>
                 dlatN=<num-expr> inN=<table> ifmtN=<in-format>
                 istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2sky in=tgas_source.fits lon=ra lat=dec
               layer1=mark
               layer2=skyvector
               dlon2=pmra dlat2=pmdec unit2=scaled arrow2=medium_arrow
               clon=56.75 clat=24.10 radius=1.5
```

`arrowN = small_arrow|medium_arrow|...` ***(MultiPointShape)***

How arrows are represented.

The available options are:

- `small_arrow`
- `medium_arrow`
- `large_arrow`
- `small_open_dart`
- `medium_open_dart`
- `large_open_dart`
- `small_filled_dart`
- `medium_filled_dart`
- `large_filled_dart`
- `lines`
- `capped_lines`

[Default: `small_arrow`]

`dlatN = <num-expr>` ***(String)***

Change in the latitude coordinate represented by the plotted vector. The units of this angular extent are determined by the `unit` option.

The value is a numeric algebraic expression based on column names as described in Section 10.

dlonN = <num-expr> ***(String)***

Change in the longitude coordinate represented by the plotted vector. The supplied value **is** considered to be premultiplied by cos(Latitude). The units of this angular extent are determined by the `unit` option.

The value is a numeric algebraic expression based on column names as described in Section 10.

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

latN = <deg-expr> ***(String)***

Latitude in decimal degrees.

The value is a numeric algebraic expression based on column names as described in Section

10.

`lonN = <deg-expr>` ***(String)***
Longitude in decimal degrees.

The value is a numeric algebraic expression based on column names as described in Section 10.

`scaleN = <number>` ***(Double)***
Scales the size of variable-sized markers like vectors and ellipses. The default value is 1, smaller or larger values multiply the visible sizes accordingly.

The main purpose of this option is to tweak the visible sizes of the plotted markers for better visibility. The `unit` option is more convenient to account for the units in which the angular extent coordinates are supplied. If the markers are supposed to be plotted with their absolute angular extents visible, this option should be set to its default value of 1.

[Default: `1`]

`shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***
Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto` (Section 8.4.1)
- `flat` (Section 8.4.2)
- `translucent` (Section 8.4.3)
- `transparent` (Section 8.4.4)
- `density` (Section 8.4.5)
- `aux` (Section 8.4.6)
- `weighted` (Section 8.4.7)

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

`thickN = <int-value>` ***(Integer)***
Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

`unitN = scaled|radian|degree|minute|arcsec|mas|uas` ***(AngleUnit)***
Defines the units in which the angular extents are specified. Options are degrees, arcseconds etc. If the special value `scaled` is given then a non-physical scaling is applied to the input values to make the the largest markers appear at a reasonable size (a few tens of pixels) in the plot.

Note that the actual plotted size of the markers can also be scaled using the `scale` option; these two work together to determine the actual plotted sizes.

The available options are:

- `scaled`: a non-physical scaling is applied based on the size of values present
- `radian`: radians
- `degree`: degrees
- `minute`: arcminutes
- `arcsec`: arcseconds
- `mas`: milli-arcseconds
- `uas`: micro-arcseconds

[Default: `degree`]

8.3.37 `skyellipse`

Plots an ellipse (or rectangle, triangle, or other similar figure) defined by two principal radii and an optional angle of rotation, the so-called position angle. This angle, if specified, is in degrees and gives the angle from the North pole towards the direction of increasing longitude on the longitude axis.

The dimensions of the plotted ellipses are given by the `ra` and `rb` coordinates. The units of these values are specified using the `unit` option. If only the relative rather than the absolute sizes are required on the plot, or if the units are not known, the special value `unit=scaled` may be used; this applies a non-physical scaling factor to make the ellipses appear at some reasonable size in the plot. When `unit=scaled` ellipses will keep approximately the same screen size during zoom operations; when one of the angular units is chosen, they will keep the same size in data coordinates.

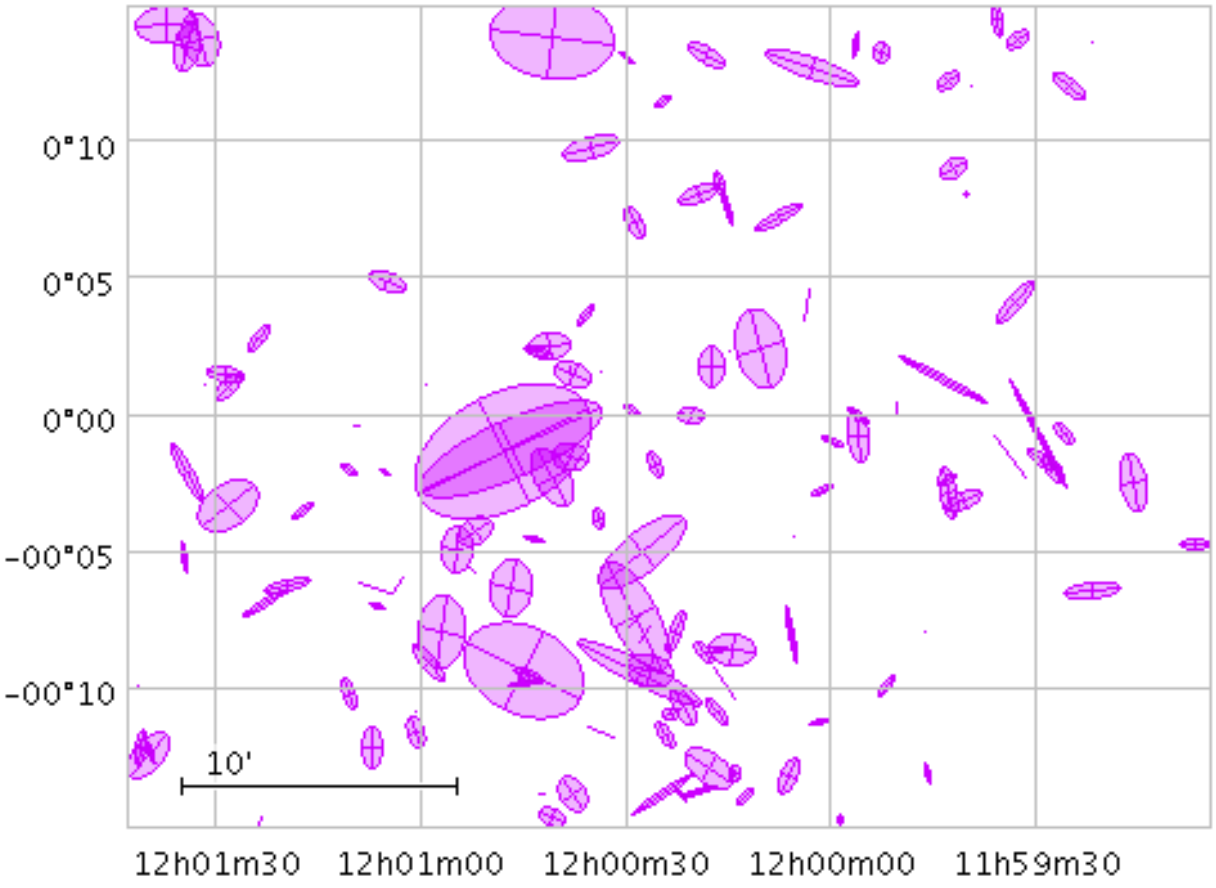
Additionally, the `scale` option may be used to scale all the plotted ellipses by a given factor to make them all larger or smaller.

Usage Overview:

```
layerN=skyellipse ellipseN=ellipse|crosshair_ellipse|... thickN=<int-value>
                  unitN=scaled|radian|degree|minute|arcsec|mas|uas
                  scaleN=<number>
                  shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-pa
                  lonN=<deg-expr> latN=<deg-expr> raN=<num-expr>
                  rbN=<num-expr> posangN=<deg-expr> inN=<table>
                  ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2sky in=mgc_ok.fits
               lon=mgc_alpha_j2000 lat=mgc_delta_j2000
               ra=bulge_re rb=bulge_re*bulge_e unit=arcsec posang=bulge_pa
               scale=10 color=#cc00ff
               layer1=skyellipse ellipse1=filled_ellipse shading1=transparent opaque1=4
               layer2=skyellipse ellipse2=crosshair_ellipse
               clon=180.1 clat=0 radius=0.25
```

`ellipseN = ellipse|crosshair_ellipse|...` ***(MultiPointShape)***

How ellipses are represented.

The available options are:

- `ellipse`
- `crosshair_ellipse`
- `filled_ellipse`
- `rectangle`
- `crosshair_rectangle`
- `filled_rectangle`
- `open_triangle`
- `filled_triangle`
- `lines`
- `capped_lines`
- `arrows`

[Default: `ellipse`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter

can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`latN = <deg-expr>` ***(String)***
Latitude in decimal degrees.

The value is a numeric algebraic expression based on column names as described in Section 10.

`lonN = <deg-expr>` ***(String)***
Longitude in decimal degrees.

The value is a numeric algebraic expression based on column names as described in Section 10.

`posangN = <deg-expr>` ***(String)***
Orientation of the ellipse. The value is the angle in degrees from the North pole to the primary axis of the ellipse in the direction of increasing longitude.

The value is a numeric algebraic expression based on column names as described in Section

10.

raN = <num-expr> ***(String)***
Ellipse first principal radius. The units of this angular extent are determined by the `unit` option.

The value is a numeric algebraic expression based on column names as described in Section 10.

rbN = <num-expr> ***(String)***
Ellipse second principal radius. The units of this angular extent are determined by the `unit` option. If this value is blank, the two radii will be assumed equal, i.e. the ellipses will be circles.

The value is a numeric algebraic expression based on column names as described in Section 10.

scaleN = <number> ***(Double)***
Scales the size of variable-sized markers like vectors and ellipses. The default value is 1, smaller or larger values multiply the visible sizes accordingly.

The main purpose of this option is to tweak the visible sizes of the plotted markers for better visibility. The `unit` option is more convenient to account for the units in which the angular extent coordinates are supplied. If the markers are supposed to be plotted with their absolute angular extents visible, this option should be set to its default value of 1.

[Default: `1`]

shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN> ***(ShapeMode)***
Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

thickN = <int-value> ***(Integer)***
Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

unitN = scaled|radian|degree|minute|arcsec|mas|uas ***(AngleUnit)***
Defines the units in which the angular extents are specified. Options are degrees, arcseconds etc. If the special value `scaled` is given then a non-physical scaling is applied to the input values to make the the largest markers appear at a reasonable size (a few tens of pixels) in the plot.

Note that the actual plotted size of the markers can also be scaled using the `scale` option; these two work together to determine the actual plotted sizes.

The available options are:

- `scaled`: a non-physical scaling is applied based on the size of values present
- `radian`: radians
- `degree`: degrees
- `minute`: arcminutes
- `arcsec`: arcseconds
- `mas`: milli-arcseconds
- `uas`: micro-arcseconds

[Default: `degree`]

8.3.38 `skycorr`

Plots an error ellipse (or rectangle or other similar figure) on the sky defined by errors in the longitude and latitude directions, and a correlation between the two errors.

The error in longitude is considered to be premultiplied by the cosine of the latitude, i.e. both errors correspond to angular distances along a great circle.

The supplied correlation is a dimensionless value in the range -1..+1 and is equal to the covariance divided by the product of the lon and lat errors. The covariance matrix is thus:

```
[  lonerr*lonerr       lonerr*laterr*corr  ]
[  lonerr*laterr*corr  laterr*laterr       ]
```

The dimensions of the plotted ellipses are given by the `lonerr` and `laterr` coordinates. The units of these values are specified using the `unit` option. If only the relative rather than the absolute sizes are required on the plot, or if the units are not known, the special value `unit=scaled` may be used; this applies a non-physical scaling factor to make the ellipses appear at some reasonable size in the plot. When `unit=scaled` ellipses will keep approximately the same screen size during zoom operations; when one of the angular units is chosen, they will keep the same size in data coordinates.

Additionally, the `scale` option may be used to scale all the plotted ellipses by a given factor to make them all larger or smaller.

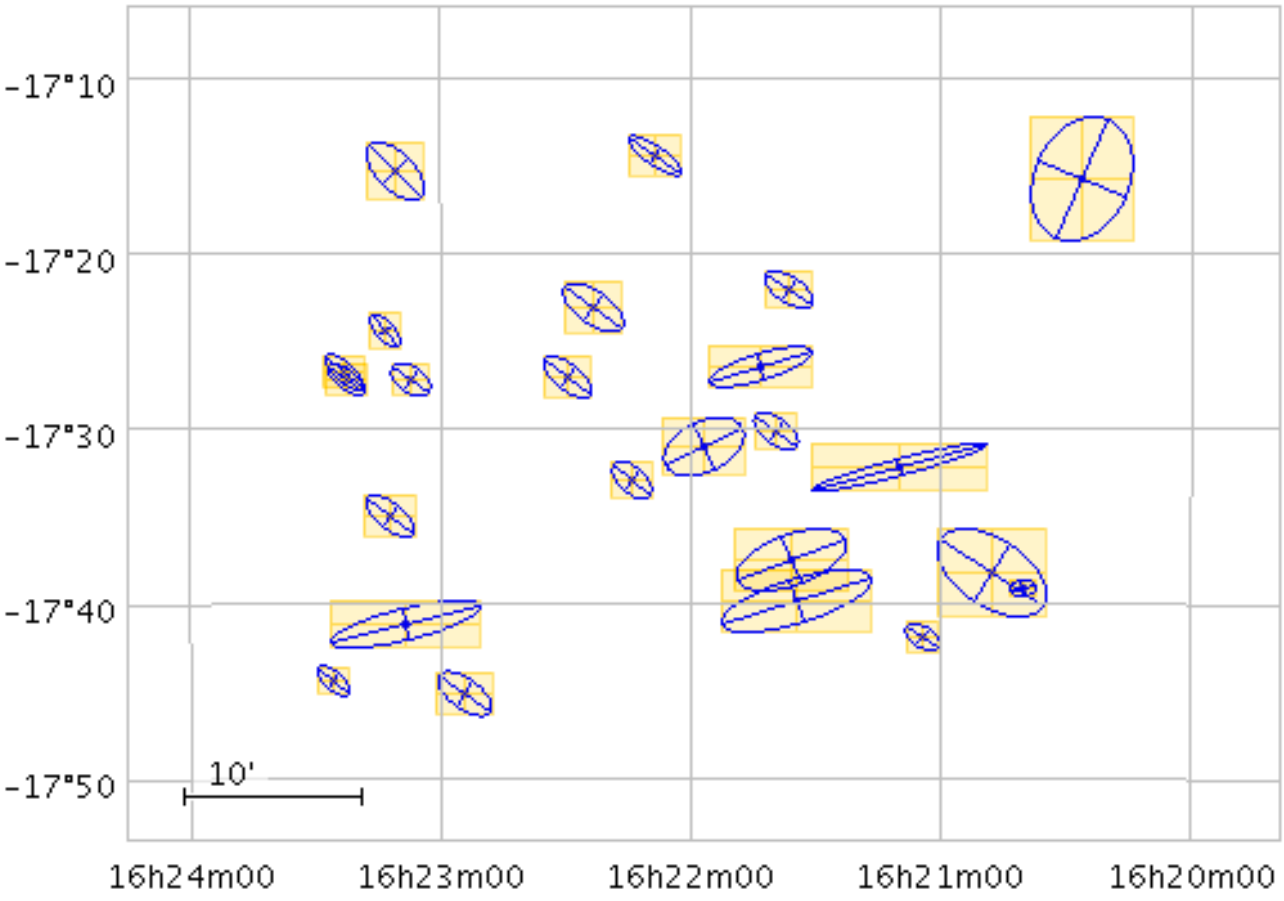
This plot type is suitable for use with the `ra_error`, `dec_error` and `ra_dec_corr` columns in the *Gaia* source catalogue. Note that Gaia positional errors are generally quoted in milli-arcseconds, so you should set `unit=mas`. Note also that in most plots Gaia positional errors are much too small to see!

Usage Overview:

```
layerN=skycorr ellipseN=ellipse|crosshair_ellipse|... thickN=<int-value>
               unitN=scaled|radian|degree|minute|arcsec|mas|uas
               scaleN=<number>
               shadingN=auto|flat|translucent|transparent|density|aux|weighted <shade-param
               lonN=<deg-expr> latN=<deg-expr> lonerrN=<num-expr>
               laterrN=<num-expr> corrN=<num-expr> inN=<table>
               ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2sky in=tgas_source.fits
               lon=ra lat=dec
               icmd='select ra>245.1&&ra<245.9&&dec>-17.8&&dec<-17.2'
               color=blue
               layer1=mark
               unit=mas scale=2e5
               ra2=ra_error rb2=dec_error posang2=90
               color2=orange shading2=transparent
               layer2a=skyellipse ellipse2a=filled_rectangle opaque2a=6
               layer2b=skyellipse ellipse2b=crosshair_rectangle opaque2b=2
               layer3=skycorr
               lonerr3=ra_error laterr3=dec_error corr3=ra_dec_corr
               ellipse3=crosshair_ellipse
```

`corrN = <num-expr>` ***(String)***

Correlation between the errors in longitude and latitude. This is a dimensionless quantity in the range -1..+1, and is equivalent to the covariance divided by the product of the Longitude and Latitude error values themselves. It corresponds to the `ra_dec_corr` value supplied in the Gaia source catalogue.

The value is a numeric algebraic expression based on column names as described in Section 10.

`ellipseN = ellipse|crosshair_ellipse|...` ***(MultiPointShape)***

How ellipses are represented.

The available options are:

- `ellipse`
- `crosshair_ellipse`
- `filled_ellipse`
- `rectangle`
- `crosshair_rectangle`

- `filled_rectangle`
- `open_triangle`
- `filled_triangle`
- `lines`
- `capped_lines`
- `arrows`

[Default: `ellipse`]

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`latN = <deg-expr>` ***(String)***
Latitude in decimal degrees.

The value is a numeric algebraic expression based on column names as described in Section 10.

laterrN = <num-expr> ***(String)***
Error in the latitude coordinate. The units of this angular extent are determined by the `unit` option.

The value is a numeric algebraic expression based on column names as described in Section 10.

lonN = <deg-expr> ***(String)***
Longitude in decimal degrees.

The value is a numeric algebraic expression based on column names as described in Section 10.

lonerrN = <num-expr> ***(String)***
Error in the longitude coordinate. The supplied value **is** considered to be premultiplied by cos(Latitude). The units of this angular extent are determined by the `unit` option.

The value is a numeric algebraic expression based on column names as described in Section 10.

scaleN = <number> ***(Double)***
Scales the size of variable-sized markers like vectors and ellipses. The default value is 1, smaller or larger values multiply the visible sizes accordingly.

The main purpose of this option is to tweak the visible sizes of the plotted markers for better visibility. The `unit` option is more convenient to account for the units in which the angular extent coordinates are supplied. If the markers are supposed to be plotted with their absolute angular extents visible, this option should be set to its default value of 1.

[Default: `1`]

shadingN = auto|flat|translucent|transparent|density|aux|weighted <shade-paramsN> ***(ShapeMode)***
Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `auto (Section 8.4.1)`
- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `auto`]

thickN = <int-value> ***(Integer)***
Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

unitN = scaled|radian|degree|minute|arcsec|mas|uas ***(AngleUnit)***
Defines the units in which the angular extents are specified. Options are degrees, arcseconds etc. If the special value `scaled` is given then a non-physical scaling is applied to the input values to make the the largest markers appear at a reasonable size (a few tens of pixels) in the

plot.

Note that the actual plotted size of the markers can also be scaled using the `scale` option; these two work together to determine the actual plotted sizes.

The available options are:

- `scaled`: a non-physical scaling is applied based on the size of values present
- `radian`: radians
- `degree`: degrees
- `minute`: arcminutes
- `arcsec`: arcseconds
- `mas`: milli-arcseconds
- `uas`: micro-arcseconds

[Default: `degree`]

8.3.39 `skydensity`

Plots a density map on the sky. The grid on which the values are drawn uses the HEALPix tesselation, with a configurable resolution. You can optionally use a weighting for the points, and you can configure how the points are combined to produce the output pixel values.

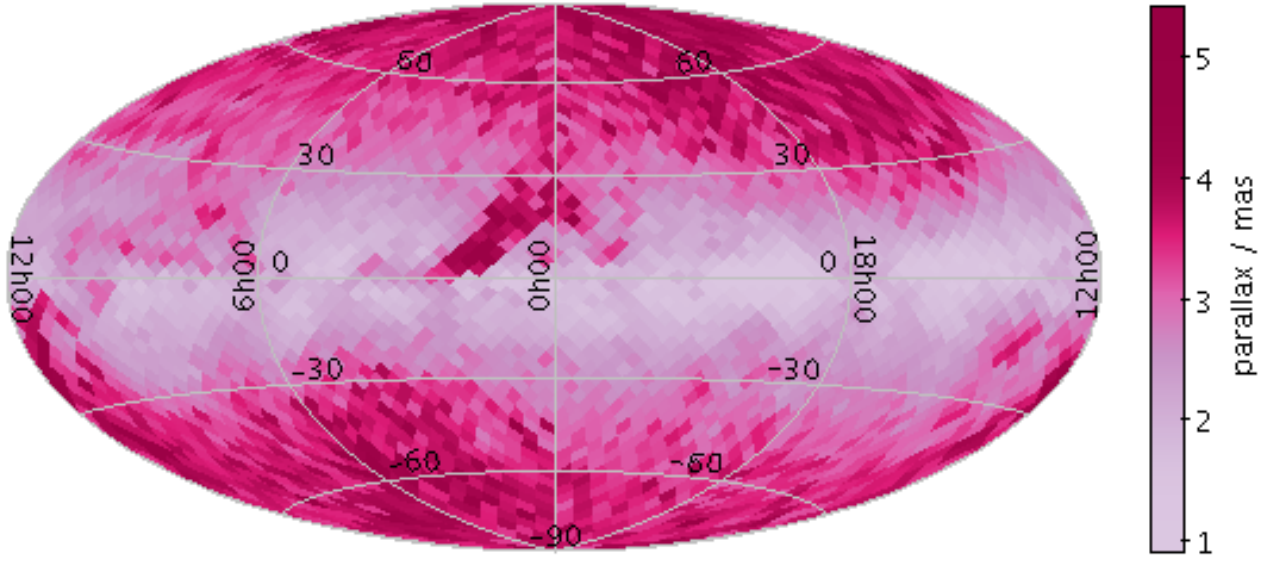
The way that data values are mapped to colours is usually controlled by options at the level of the plot itself, rather than by per-layer configuration.

Usage Overview:

```
layerN=skydensity levelN=<-rel-level|+abs-level>
                  combineN=sum|sum-per-unit|count|...
                  perunitN=steradian|degree2|arcmin2|arcsec2|mas2|uas2
                  transparencyN=0..1 lonN=<deg-expr> latN=<deg-expr>
                  weightN=<num-expr> inN=<table> ifmtN=<in-format>
                  istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2sky in=tgas_source.fits lon=l lat=b
```

```
layer1=skydensity weight1=parallax combine1=mean level1=4
projection=aitoff auxmap=PuRd auxfunc=histogram
xpix=540 ypix=250
```

`combineN = sum|sum-per-unit|count|...` ***(Combiner)***

Defines how values contributing to the same density map bin are combined together to produce the value assigned to that bin, and hence its colour. The combined values are the weights, but if the `weight` is left blank, a weighting of unity is assumed.

For density-like values (`count-per-unit`, `sum-per-unit`) the scaling is additionally influenced by the `perunit` parameter.

The available options are:

- `sum`: the sum of all the combined values per bin
- `sum-per-unit`: the sum of all the combined values per unit of bin size
- `count`: the number of non-blank values per bin (weight is ignored)
- `count-per-unit`: the number of non-blank values per unit of bin size (weight is ignored)
- `mean`: the mean of the combined values
- `median`: the median
- `q1`: first quartile
- `q3`: third quartile
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `stdev`: the sample standard deviation of the combined values
- `hit`: 1 if any values present, NaN otherwise (weight is ignored)

[Default: `sum-per-unit`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.

- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`latN = <deg-expr>` ***(String)***

Latitude in decimal degrees.

The value is a numeric algebraic expression based on column names as described in Section 10.

`levelN = <-rel-level|+abs-level>` ***(Integer)***

Determines the HEALPix level of pixels which are averaged over to calculate density.

If the supplied value is a non-negative integer, it gives the absolute level to use; at level 0 there are 12 pixels on the sky, and the count multiplies by 4 for each increment.

If the value is negative, it represents a relative level; it is approximately the (negative) number of screen pixels along one side of a HEALPix sky pixel. In this case the actual HEALPix level will depend on the current zoom.

[Default: `-3`]

`lonN = <deg-expr>` ***(String)***

Longitude in decimal degrees.

The value is a numeric algebraic expression based on column names as described in Section 10.

`perunitN = steradian|degree2|arcmin2|arcsec2|mas2|uas2` ***(SolidAngleUnit)***

Defines the unit of sky area used for scaling density-like combinations (e.g. `combine=count-per-unit` or `sum-per-unit`). If the Combination mode is calculating values per unit area, this configures the area scale in question. For non-density-like combination modes (e.g. `combine=sum` or `mean`) it has no effect.

The available options are:

- `steradian`: (180/Pi)^2 deg^2
- `degree2`: square degrees
- `arcmin2`: square arcminute, (1/60)^2 deg^2
- `arcsec2`: square arcsecond, (1/3600)^2 deg^2
- `mas2`: square milliarcsecond, (.001/3600)^2 deg^2
- `uas2`: square microarcsecond, (1e-6/3600)^2 deg^2

[Default: `degree2`]

`transparencyN = 0..1` ***(Double)***

Transparency with which components are plotted, in the range 0 (opaque) to 1 (invisible). The value is 1-alpha.

[Default: `0`]

`weightN = <num-expr>` ***(String)***

Weighting of data points. If supplied, each point contributes a value to the histogram equal to the data value multiplied by this coordinate. If not supplied, the effect is the same as supplying a fixed value of one.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.40 `healpix`

Plots a table representing HEALPix pixels on the sky. Each row represents a single HEALPix tile, and a value from that row is used to colour the corresponding region of the sky plot.

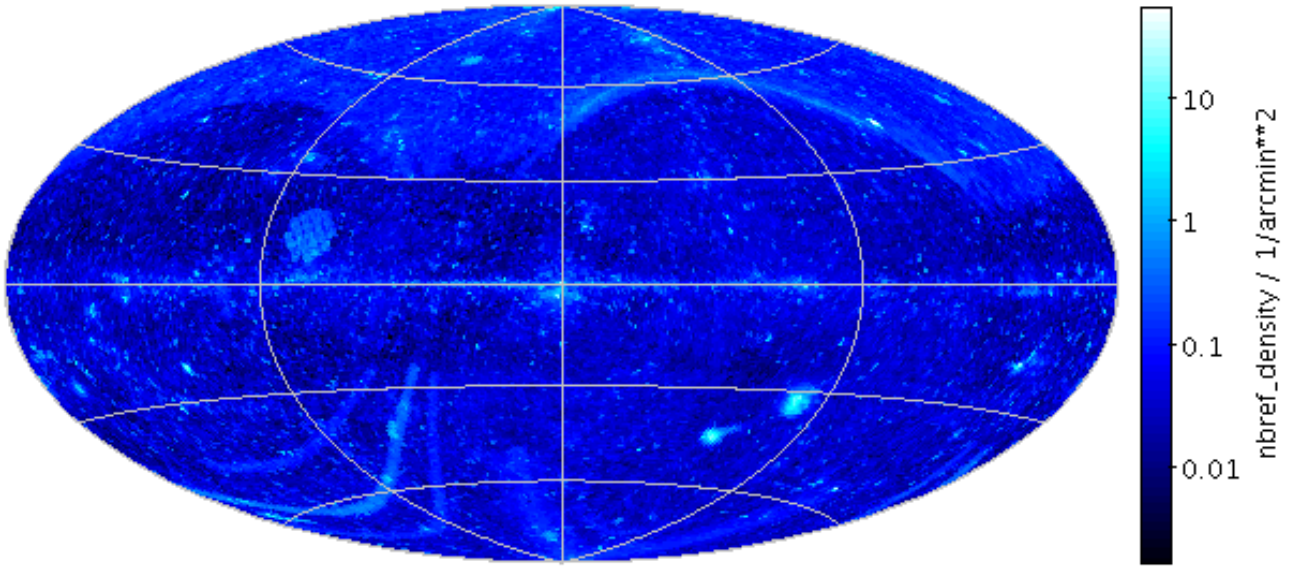
The way that data values are mapped to colours is usually controlled by options at the level of the plot itself, rather than by per-layer configuration.

Usage Overview:

```
layerN=healpix datalevelN=<int-value> datasysN=<value>
               viewsysN=equatorial|galactic|supergalactic|ecliptic
               degradeN=<int-value> combineN=sum|sum-per-unit|count|...
               perunitN=steradian|degree2|arcmin2|arcsec2|mas2|uas2
               transparencyN=0..1 healpixN=<num-expr> valueN=<num-expr>
               inN=<table> ifmtN=<in-format> istreamN=true|false
               icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2sky layer1=healpix in1=simbad-hpx8.fits healpix1=HPX8 value1=NBREF
                datalevel1=8 degrade1=2 combine=sum-per-unit perunit=arcmin2
                projection=aitoff datasys1=equatorial viewsys=galactic labelpos=none
                auxfunc=log auxmap=cold auxflip=true auxclip=0,1
                xpix=600 ypix=280
```

`combineN = sum|sum-per-unit|count|...` ***(Combiner)***

Defines how values degraded to a lower HEALPix level are combined together to produce the

value assigned to the larger tile, and hence its colour. This is mostly useful in the case that `degrade`>0.

For density-like values (`count-per-unit`, `sum-per-unit`) the scaling is additionally influenced by the `perunit` parameter.

The available options are:

- `sum`: the sum of all the combined values per bin
- `sum-per-unit`: the sum of all the combined values per unit of bin size
- `count`: the number of non-blank values per bin (weight is ignored)
- `count-per-unit`: the number of non-blank values per unit of bin size (weight is ignored)
- `mean`: the mean of the combined values
- `median`: the median
- `q1`: first quartile
- `q3`: third quartile
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `stdev`: the sample standard deviation of the combined values
- `hit`: 1 if any values present, NaN otherwise (weight is ignored)

[Default: `mean`]

`datalevelN = <int-value>` ***(Integer)***
HEALPix level of the (implicit or explicit) tile indices. Legal values are between 0 (12 pixels) and 29 (3458764513820540928 pixels). If a negative value is supplied (the default), then an attempt is made to determine the correct level from the data.

[Default: `-1`]

`datasysN = <value>` ***(SkySys)***
The sky coordinate system to which the HEALPix grid used by the input pixel file refers.

[Default: `equatorial`]

`degradeN = <int-value>` ***(Integer)***
Allows the HEALPix grid to be drawn at a less detailed level than the level at which the input data are supplied. A value of zero (the default) means that the HEALPix tiles are painted with the same resolution as the input data, but a higher value will degrade resolution of the plot tiles; each plotted tile will correspond to 4^`degrade` input tiles. The way that values are combined within each painted tile is controlled by the `combine` value.

[Default: `0`]

`healpixN = <num-expr>` ***(String)***
HEALPix index indicating the sky position of the tile whose value is plotted. If not supplied, the assumption is that the supplied table contains one row for each HEALPix tile at a given level, in ascending order.

The value is a numeric algebraic expression based on column names as described in Section 10.

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter

commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`perunitN = steradian|degree2|arcmin2|arcsec2|mas2|uas2` ***(SolidAngleUnit)***

Defines the unit of sky area used for scaling density-like combinations (e.g. `combine=count-per-unit` or `sum-per-unit`). If the Combination mode is calculating values per unit area, this configures the area scale in question. For non-density-like combination modes (e.g. `combine=sum` or `mean`) it has no effect.

The available options are:

- `steradian`: (180/Pi)^2 deg^2
- `degree2`: square degrees
- `arcmin2`: square arcminute, (1/60)^2 deg^2
- `arcsec2`: square arcsecond, (1/3600)^2 deg^2
- `mas2`: square milliarcsecond, (.001/3600)^2 deg^2
- `uas2`: square microarcsecond, (1e-6/3600)^2 deg^2

[Default: `degree2`]

`transparencyN = 0..1` ***(Double)***

Transparency with which components are plotted, in the range 0 (opaque) to 1 (invisible). The value is 1-alpha.

[Default: `0`]

`valueN = <num-expr>` ***(String)***
Value of HEALPix tile, determining the colour which will be plotted.

The value is a numeric algebraic expression based on column names as described in Section 10.

`viewsysN = equatorial|galactic|supergalactic|ecliptic` ***(SkySys)***
The sky coordinate system used for the generated plot.

Choice of this value goes along with the data coordinate system that may be specified for plot layers. If unspecified, a generic longitude/latitude system is used, and all lon/lat coordinates in the plotted data layers are assumed to be in the same system. If a value is supplied for this parameter, then a sky system must (implicitly or explicitly) be supplied for each data layer, and the coordinates are converted from data to view system before being plotted.

The available options are:

- `equatorial`: J2000 equatorial system
- `galactic`: IAU 1958 galactic system
- `supergalactic`: De Vaucouleurs supergalactic system
- `ecliptic`: ecliptic system based on conversion at 2000.0

[Default: `equatorial`]

8.3.41 `skygrid`

Plots an additional axis grid on the celestial sphere. This can be overlaid on the default sky axis grid so that axes for multiple sky coordinate systems are simultaneously visible. The plots are done relative to the View sky system (`viewsys` parameter) defined for the plot as a whole.

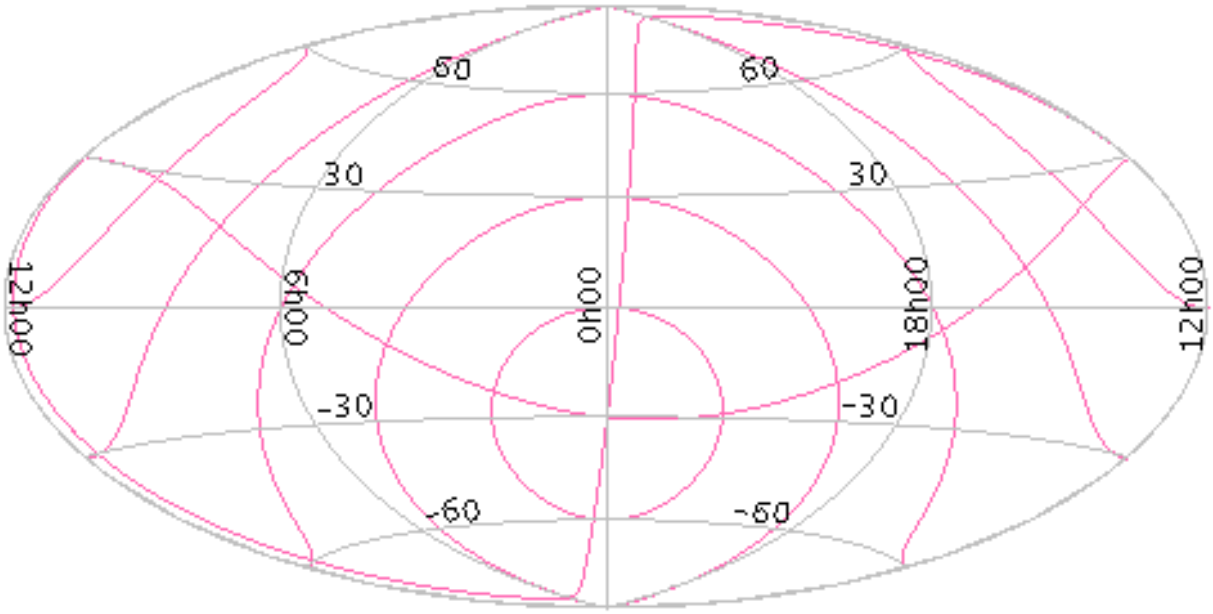
Note that some of the configuration items for this plotter, such as grid line antialiasing and the decimal/sexagesimal flag, are inherited from the values set for the main sky plot grid.

Usage Overview:

```
layerN=skygrid gridsysN=equatorial|galactic|supergalactic|ecliptic
               gridcolorN=<rrggbb>|red|blue|... transparencyN=0..1
               labelposN=Internal|None loncrowdN=<number> latcrowdN=<number>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2sky xpix=500 ypix=250 projection=aitoff
               viewsys=ecliptic
               layer1=skygrid gridsys1=galactic gridcolor1=HotPink labelpos1=none
```

`gridcolorN = <rrggbb>|red|blue|...` ***(Color)***

The color of the plot grid, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `light_grey`]

`gridsysN = equatorial|galactic|supergalactic|ecliptic` ***(SkySys)***

The sky coordinate system used for the additional grid axes. This is used in conunction with the `viewsys` parameter defined for the plot as a whole to determin what grid lines to plot.

The available options are:

- `equatorial`: J2000 equatorial system
- `galactic`: IAU 1958 galactic system
- `supergalactic`: De Vaucouleurs supergalactic system
- `ecliptic`: ecliptic system based on conversion at 2000.0

[Default: `equatorial`]

`labelposN = Internal|None` ***(SkyAxisLabeller)***

Controls how and whether the numeric annotations of the lon/lat axes are displayed.

The available options are:

- `Internal`: Labels are drawn inside the plot bounds
- `None`: Axes are not labelled

[Default: `Internal`]

`latcrowdN = <number>` ***(Double)***
Determines how closely sky latitude grid lines (parallels) are spaced. The default value is 1, meaning normal crowding. Larger values result in more grid lines, and smaller values in fewer grid lines.

[Default: `1`]

`loncrowdN = <number>` ***(Double)***
Determines how closely sky longitude grid lines (meridians) are spaced. The default value is 1, meaning normal crowding. Larger values result in more grid lines, and smaller values in fewer grid lines.

[Default: `1`]

`transparencyN = 0..1` ***(Double)***
Transparency with which components are plotted, in the range 0 (opaque) to 1 (invisible). The value is 1-alpha.

[Default: `0`]

8.3.42 `xyzvector`

Plots directed lines from the data position given delta values for the coordinates. The plotted markers are typically little arrows, but there are other options.

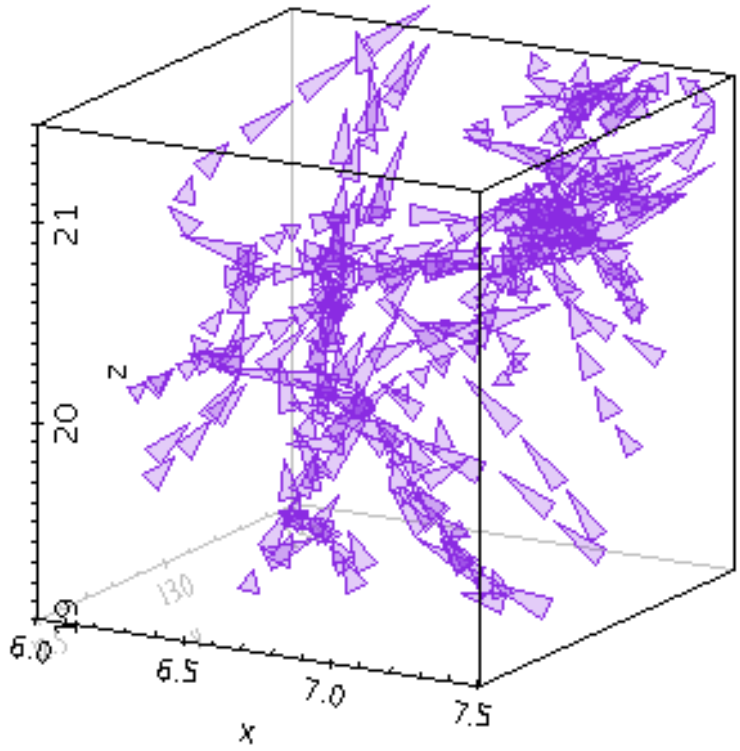
In some cases the supplied data values give the actual extents in data coordinates for the plotted vectors but sometimes the data is on a different scale or in different units to the positional coordinates. As a convenience for this case, the plotter can optionally scale the magnitudes of all the vectors to make them a reasonable size on the plot, so by default the largest ones are a few tens of pixels long. This auto-scaling is turned off by default, but it can be activated with the `autoscale` option. Whether autoscaling is on or off, the `scale` option can be used to apply a fixed scaling factor.

Usage Overview:

```
layerN=xyzvector arrowN=small_arrow|medium_arrow|... thickN=<int-value>
                 scaleN=<factor> autoscaleN=true|false
                 shadingN=flat|translucent|transparent|density|aux|weighted <shade-paramsN>
                 xN=<num-expr> yN=<num-expr> zN=<num-expr>
                 xdeltaN=<num-expr> ydeltaN=<num-expr> zdeltaN=<num-expr>
                 inN=<table> ifmtN=<in-format> istreamN=true|false
                 icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2cube in=gavo_g2.fits
                 x=x y=y z=z xdelta=velX ydelta=velY zdelta=velZ autoscale=true
                 color=BlueViolet scale=1.5
                 layer1=xyzvector shading1=transparent opaque1=5 arrow1=medium_filled_dart
                 layer2=xyzvector shading2=flat arrow2=medium_open_dart
                 xmin=6 xmax=7.5 ymin=12.5 ymax=13.5 zmin=19 zmax=21.5
```

`arrowN = small_arrow|medium_arrow|...` ***(MultiPointShape)***

How arrows are represented.

The available options are:

- `small_arrow`
- `medium_arrow`
- `large_arrow`
- `small_open_dart`
- `medium_open_dart`
- `large_open_dart`
- `small_filled_dart`
- `medium_filled_dart`
- `large_filled_dart`
- `lines`
- `capped_lines`

[Default: `small_arrow`]

`autoscaleN = true|false` ***(Boolean)***

Determines whether the default size of variable-sized markers like vectors and ellipses are automatically scaled to have a sensible size. If true, then the sizes of all the plotted markers are examined, and some dynamically calculated factor is applied to them all to make them a sensible size (by default, the largest ones will be a few tens of pixels). If false, the sizes will be

the actual input values interpreted in data coordinates.

If auto-scaling is on, then markers will keep approximately the same screen size during zoom operations; if it's off, they will keep the same size in data coordinates.

Marker size is also affected by the `scale` parameter.

[Default: `false`]

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`scaleN = <factor>` ***(Double)***

Affects the size of variable-sized markers like vectors and ellipses. The default value is 1, smaller or larger values multiply the visible sizes accordingly.

[Default: `1`]

`shadingN = flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `flat`]

`thickN = <int-value>` ***(Integer)***

Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

`xN = <num-expr>` ***(String)***

X coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xdeltaN = <num-expr>` ***(String)***

Vector component in the X direction.

The value is a numeric algebraic expression based on column names as described in Section 10.

`yN = <num-expr>` ***(String)***

Y coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`ydeltaN = <num-expr>` ***(String)***

Vector component in the Y direction.

The value is a numeric algebraic expression based on column names as described in Section 10.

`zN = <num-expr>` ***(String)***

Z coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`zdeltaN = <num-expr>` ***(String)***

Vector component in the Z direction.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.43 `xyzerror`

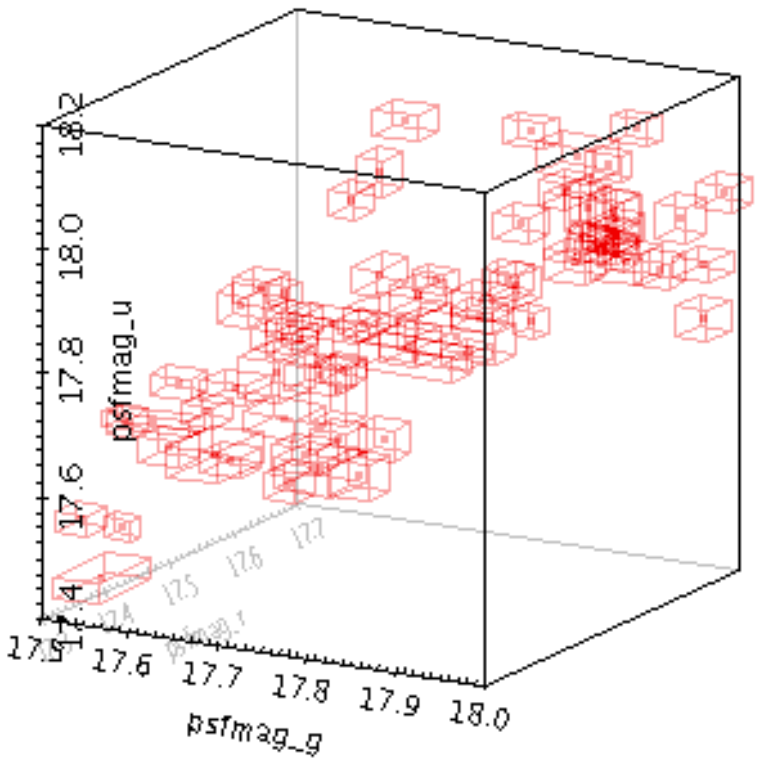
Plots symmetric or asymmetric error bars in some or all of the plot dimensions. The shape of the error "bars" is quite configurable, including (for 2-d and 3-d errors) ellipses, rectangles etc aligned with the axes.

Usage Overview:

```
layerN=xyzerror errorbarN=none|lines|capped_lines|... thickN=<int-value>
                shadingN=flat|translucent|transparent|density|aux|weighted <shade-paramsN>
                xN=<num-expr> yN=<num-expr> zN=<num-expr> xerrhiN=<num-expr>
                xerrloN=<num-expr> yerrhiN=<num-expr> yerrloN=<num-expr>
                zerrhiN=<num-expr> zerrloN=<num-expr> inN=<table>
                ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2cube in=dr5qso.fits icmd='select morphology==1'
                 x=psfmag_g xerrhi=psfmagerr_g
                 y=psfmag_r yerrhi=psfmagerr_r
                 z=psfmag_u zerrhi=psfmagerr_u
                 layer1=mark
                 layer2=xyzerror errorbar2=cuboid
                 shading=transparent opaque=3
                 xmin=17.5 xmax=18 ymin=17.3 ymax=17.7 zmin=17.4 zmax=18.2
```

`errorbarN = none|lines|capped_lines|...` ***(MultiPointShape)***

How errorbars are represented.

The available options are:

- none

- `lines`
- `capped_lines`
- `caps`
- `arrows`
- `cuboid`
- `ellipse`
- `crosshair_ellipse`
- `rectangle`
- `crosshair_rectangle`
- `filled_ellipse`
- `filled_rectangle`

[Default: `lines`]

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This

parameter is ignored for scheme-specified tables.

[Default: `false`]

`shadingN = flat|translucent|transparent|density|aux|weighted <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `flat (Section 8.4.2)`
- `translucent (Section 8.4.3)`
- `transparent (Section 8.4.4)`
- `density (Section 8.4.5)`
- `aux (Section 8.4.6)`
- `weighted (Section 8.4.7)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `flat`]

`thickN = <int-value>` ***(Integer)***

Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

`xN = <num-expr>` ***(String)***

X coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xerrhiN = <num-expr>` ***(String)***

Error in the X coordinate in the positive direction. If no corresponding negative error value is supplied, then this value is also used in the negative direction, i.e. in that case errors are assumed to be symmetric.

The value is a numeric algebraic expression based on column names as described in Section 10.

`xerrloN = <num-expr>` ***(String)***

Error in the X coordinate in the negative direction. If left blank, it is assumed to take the same value as the positive error.

The value is a numeric algebraic expression based on column names as described in Section 10.

`yN = <num-expr>` ***(String)***

Y coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`yerrhiN = <num-expr>` ***(String)***

Error in the Y coordinate in the positive direction. If no corresponding negative error value is supplied, then this value is also used in the negative direction, i.e. in that case errors are assumed to be symmetric.

The value is a numeric algebraic expression based on column names as described in Section 10.

`yerrloN = <num-expr>` ***(String)***

Error in the Y coordinate in the negative direction. If left blank, it is assumed to take the same value as the positive error.

The value is a numeric algebraic expression based on column names as described in Section 10.

`zN = <num-expr>` ***(String)***
Z coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

`zerrhiN = <num-expr>` ***(String)***
Error in the Z coordinate in the positive direction. If no corresponding negative error value is supplied, then this value is also used in the negative direction, i.e. in that case errors are assumed to be symmetric.

The value is a numeric algebraic expression based on column names as described in Section 10.

`zerrloN = <num-expr>` ***(String)***
Error in the Z coordinate in the negative direction. If left blank, it is assumed to take the same value as the positive error.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.44 `line3d`

Plots a point-to-point line joining up the positions of data points in three dimensions. There are additional options to pre-sort the points by a given quantity before drawing the lines (using the `sort` value), and to vary the colour of the line along its length (using the `aux` value). The options for controlling the Aux colour map are controlled at the level of the plot itself, rather than by per-layer configuration.

Note that the line positioning in 3d and the line segment aux colouring is somewhat approximate. In most cases it is good enough for visual inspection, but pixel-level examination may reveal discrepancies.

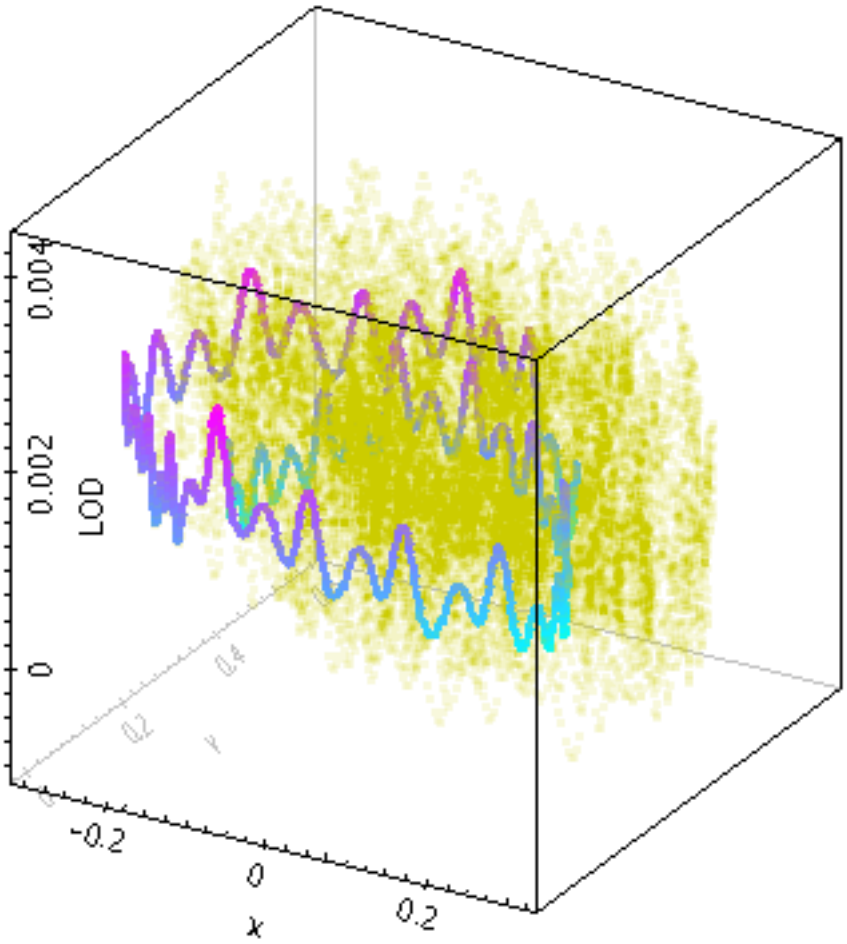
Usage Overview:

```
layerN=line3d colorN=<rrggbb>|red|blue|... thickN=<pixels>
              <pos-coord-paramsN> auxN=<num-expr> sortN=<num-expr>
              inN=<table> ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Positional Coordinate Parameters:
The positional coordinates `<pos-coord-paramsN>` give a position for each row of the input table. Their form depends on the plot geometry, i.e. which plotting command is used. For a plane plot (`plot2plane`) the parameters would be `xN` and `yN`. The coordinate parameter values are in all cases strings interpreted as numeric expressions based on column names. These can be column names, fixed values or algebraic expressions as described in Section 10.

Example:




```
stilts plot2cube in=iers.fits x=x y=y z=LOD
                layer1=line3d icmd1='select decYear>1963&&decYear<1964.5' thick1=3 aux1=LO
                layer2=mark shading2=translucent color2=cccc00 translevel2=0.35
                auxmap=cyan-magenta auxvisible=false legend=false
                phi=-150 theta=25 psi=180 xpix=400 ypix=400
```

auxN = <num-expr> ***(String)***

If supplied, this adjusts the colouring of the line along its length according to the value of this coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

colorN = <rrggbb>|red|blue|... ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

inN = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istreamN = true|false ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

sortN = <num-expr> ***(String)***

If supplied, this gives a value to define in what order points are joined together. If no value is given, the natural order is used, i.e. the sequence of rows in the table.

Note that if the required order is in fact the natural order of the table, it is better to leave this value blank, since sorting is a potentially expensive step.

The value is a numeric algebraic expression based on column names as described in Section 10.

thickN = <pixels> ***(Integer)***

Thickness of plotted line in pixels.

[Default: 1]

8.3.45 spheregrid

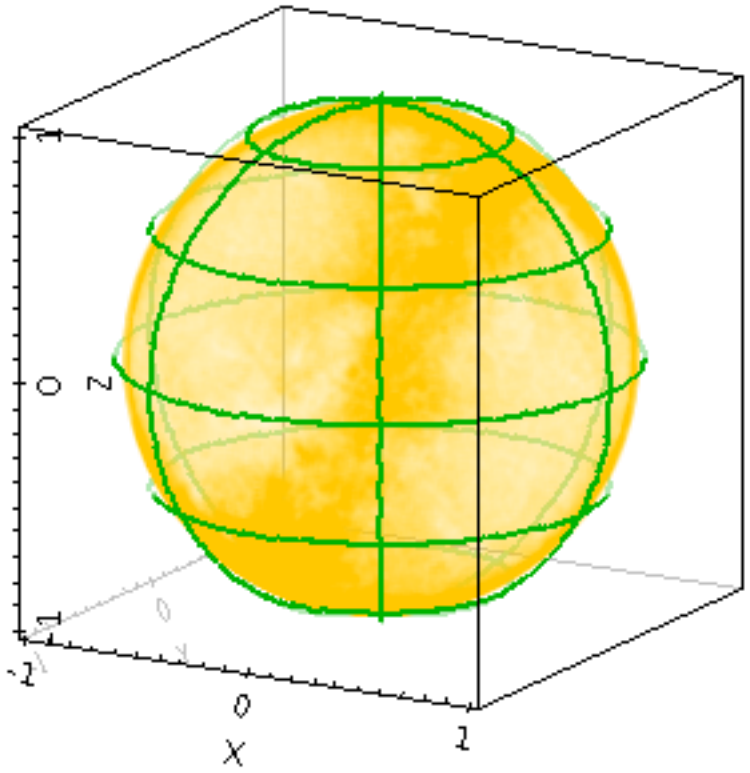
Plots a spherical grid around the origin of a 3-d plot. The radius of the sphere can be configured explicitly, otherwise a suitable default value (that should make at least some of the grid visible) will be chosen.

Usage Overview:

```
layerN=spheregrid radiusN=<number> gridcolorN=<rrggbb>|red|blue|...
                  thickN=<pixels> nlonN=<int-value> nlatN=<int-value>
```

All the parameters listed here affect only the relevant layer, identified by the suffix N.

Example:




```
stilts plot2sphere legend=false xpix=350 ypix=350
                   layer1=mark in1=tgas_source.fits lon1=ra lat1=dec r1=1
                   shading1=transparent opaque1=850 color1=orange
                   layer2=spheregrid gridcolor2=green thick2=2
```

gridcolorN = <rrggbb>|red|blue|... ***(Color)***

The color of the spherical grid, given by name or as a hexadecimal RGB value.

The standard plotting colour names are red, blue, green, grey, magenta, cyan, orange, pink, yellow, black, light_grey, white. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by

most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `grey`]

`nlatN = <int-value>` ***(Integer)***
Number of latitude lines to plot both above and below the equator in the spherical grid. A value of zero plots just the equator, and negative values plot no parallels at all.

[Default: `2`]

`nlonN = <int-value>` ***(Integer)***
Number of longitude great circles to plot in the spherical grid. Since each great circle joins the poles in two hemispheres, this value is half the number of meridians to be drawn.

[Default: `3`]

`radiusN = <number>` ***(Double)***
Defines the radius of the spherical grid that is drawn around the origin. Positive values give the radius in data units, negative values provide a multiplier for the default radius which is chosen on the basis of the currently visible volume.

[Default: `-1`]

`thickN = <pixels>` ***(Integer)***
Thickness of plotted line in pixels.

[Default: `1`]

8.3.46 `yerror`

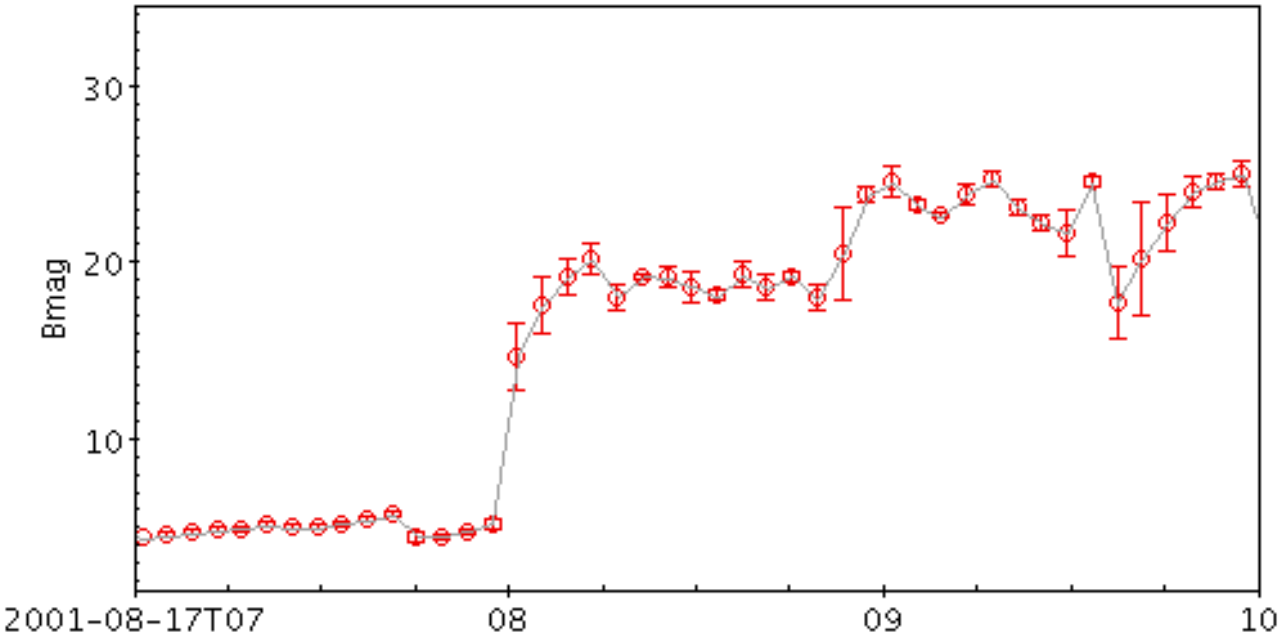
Plots symmetric or asymmetric error bars in the Y direction.

Usage Overview:

```
layerN=yerror errorbarN=none|lines|capped_lines|caps|arrows
              thickN=<int-value> shadingN=flat <shade-paramsN>
              tN=<time-expr> ttypeN=DecYear|MJD|JD|Unix|Iso8601
              yN=<num-expr> yerrhiN=<num-expr> yerrloN=<num-expr>
              inN=<table> ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2time in=ACE_data.vot t=epoch y=Bmag
                layer1=yerror yerrhi1=sigma_B errorbar1=capped_lines
                layer2=mark shape2=open_circle size2=3
                layer3=line color3=a0a0a0
                tmin=2001-08-17T07 tmax=2001-08-17T10 ypix=250
```

errorbarN = none|lines|capped_lines|caps|arrows ***(MultiPointShape)***

How errorbars are represented.

The available options are:

- `none`
- `lines`
- `capped_lines`
- `caps`
- `arrows`

[Default: `lines`]

icmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ifmtN = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`shadingN = flat <shade-paramsN>` ***(ShapeMode)***

Determines how plotted objects in layer N are coloured. This may be influenced by how many objects are plotted over each other as well as the values of other parameters. Available options (Section 8.4) are:

- `flat (Section 8.4.2)`

Each of these options comes with its own set of parameters to specify the details of how colouring is done.

[Default: `flat`]

`tN = <time-expr>` ***(String)***

Time coordinate.

The value is a Time value algebraic expression based on column names as described in Section 10.

`thickN = <int-value>` ***(Integer)***

Controls the line thickness used when drawing shapes. Zero, the default value, means a 1-pixel-wide line is used. Larger values make drawn lines thicker, but note changing this value will not affect all shapes, for instance filled rectangles contain no line drawings.

[Default: `0`]

`ttypeN = DecYear|MJD|JD|Unix|Iso8601` ***(TimeMapper)***

Selects the form in which the Time value for parameter `tN` is supplied. Options are:

- `DecYear`: Years since 0 AD
- `MJD`: Modified Julian Date
- `JD`: Julian Day
- `Unix`: Seconds since midnight 1 Jan 1970
- `Iso8601`: ISO 8601 string

If left blank, a guess will be taken depending on the data type of the value supplied for the `tN`

value.

yN = <num-expr> ***(String)***
Vertical coordinate.

The value is a numeric algebraic expression based on column names as described in Section 10.

yerrhiN = <num-expr> ***(String)***
Error in the Y coordinate in the positive direction. If no corresponding negative error value is supplied, then this value is also used in the negative direction, i.e. in that case errors are assumed to be symmetric.

The value is a numeric algebraic expression based on column names as described in Section 10.

yerrloN = <num-expr> ***(String)***
Error in the Y coordinate in the negative direction. If left blank, it is assumed to take the same value as the positive error.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.3.47 `spectrogram`

Plots spectrograms. A spectrogram is a sequence of spectra plotted as vertical 1-d images, each one plotted at a different horizontal coordinate.

This specialised layer is only available for `time` plots.

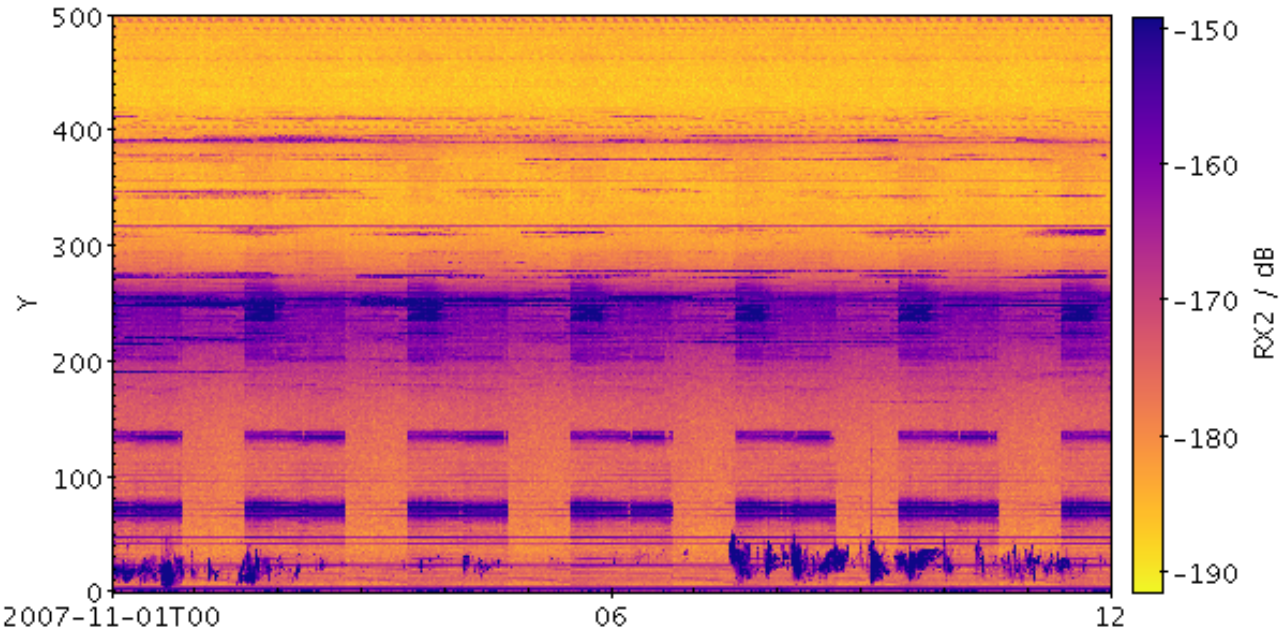
The way that data values are mapped to colours is usually controlled by options at the level of the plot itself, rather than by per-layer configuration.

Usage Overview:

```
layerN=spectrogram tN=<time-expr> ttypeN=DecYear|MJD|JD|Unix|Iso8601
                   spectrumN=<array-expr> twidthN=<num-expr> inN=<table>
                   ifmtN=<in-format> istreamN=true|false icmdN=<cmds>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2time layer1=spectrogram in1=LRS_NPW_V010_20071101.cdf t1=epoch spectrum1=RX2
                auxfunc=linear auxmap=plasma auxclip=0,1
                xpix=600 ypix=300
                tmin=2007-11-01T00 tmax=2007-11-01T12 ymax=500
```

`icmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the layer N input table as specified by parameter `inN`. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its

standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`spectrumN = <array-expr>` ***(String)***
Provides an array of spectral samples at each data point. The value must be a numeric array (e.g. the value of an array-valued column).

The value is an array-valued algebraic expression based on column names as described in Section 10. Some of the functions in the Arrays class may be useful here.

`tN = <time-expr>` ***(String)***
Time coordinate.

The value is a Time value algebraic expression based on column names as described in Section 10.

`ttypeN = DecYear|MJD|JD|Unix|Iso8601` ***(TimeMapper)***
Selects the form in which the Time value for parameter `tN` is supplied. Options are:

- `DecYear`: Years since 0 AD
- `MJD`: Modified Julian Date
- `JD`: Julian Day
- `Unix`: Seconds since midnight 1 Jan 1970
- `Iso8601`: ISO 8601 string

If left blank, a guess will be taken depending on the data type of the value supplied for the `tN` value.

`twidthN = <num-expr>` ***(String)***
Range on the Time axis over which the spectrum is plotted. If no value is supplied, an attempt will be made to determine it automatically by looking at the spacing of the Time coordinates plotted in the spectrogram.

The value is a numeric algebraic expression based on column names as described in Section 10.

8.4 Shading Modes

Some plot layer types have an associated `shading` parameter which determines how plotted markers are coloured. This is independent of the marker shapes (which may be points, vectors, ellipses, ...) but may be affected by how many markers are plotted on top of each other, additional input table values, selected colour maps etc. For the simplest shading types (e.g. `flat`) it's just a case of choosing a colour, but the more complex ones have several associated parameters.

The various shading types and their usages are described in the following subsections.

8.4.1 `auto`

Paints isolated points in their selected colour but where multiple points *in the same layer* overlap it adjusts the clour by darkening it. This means that for isolated points (most or all points in a non-crowded plot, or outliers in a crowded plot) it behaves just like flat mode, but it's easy to see where overdense regions lie.

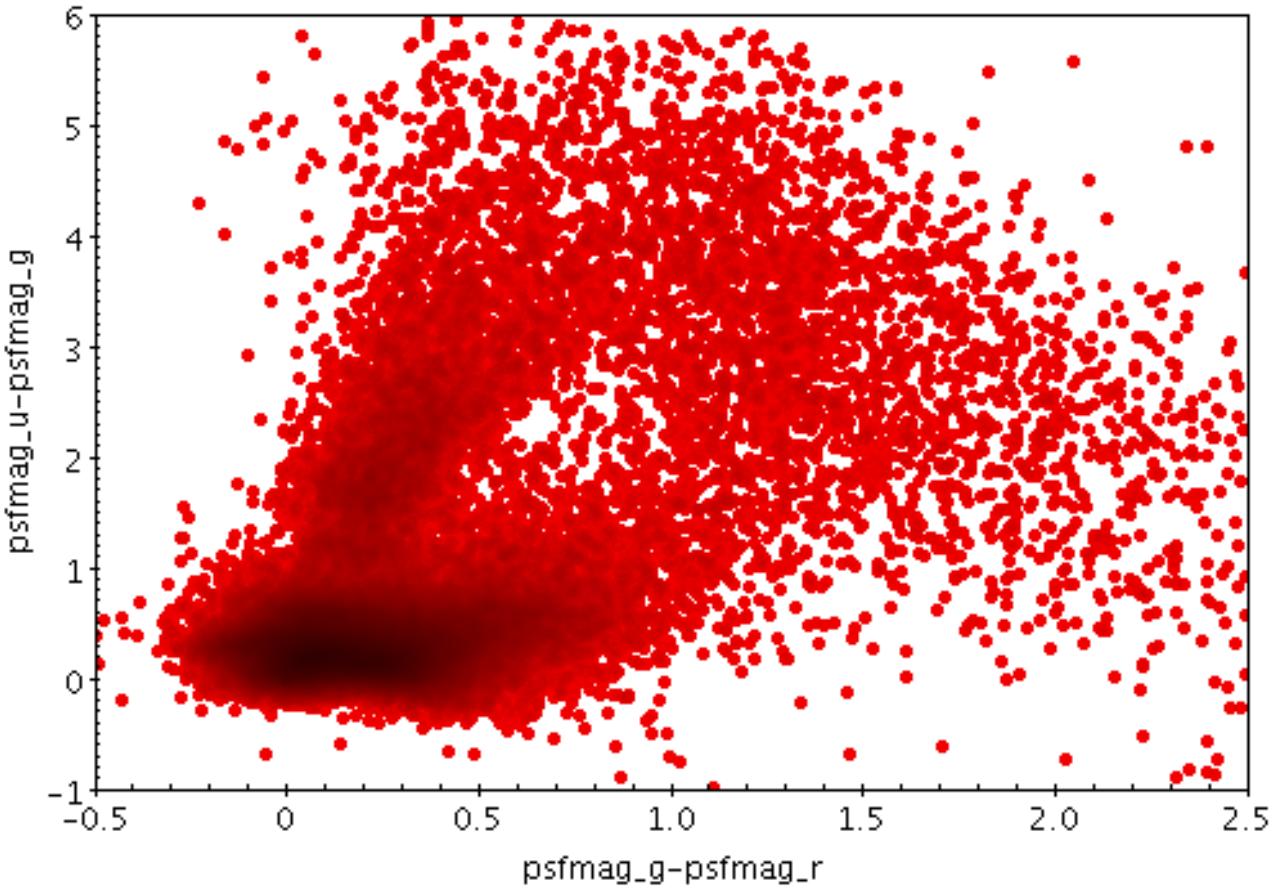
This is like density mode, but with no user-configurable options.

Usage:

```
shadingN=auto colorN=<rrggbb>|red|blue|...
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane layer1=mark in1=dr5qso.fits
                  shading1=auto
                  x1=psfmag_g-psfmag_r y1=psfmag_u-psfmag_g size1=2
                  xmin=-0.5 xmax=2.5 ymin=-1 ymax=6
```

Associated parameters are as follows:

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

8.4.2 `flat`

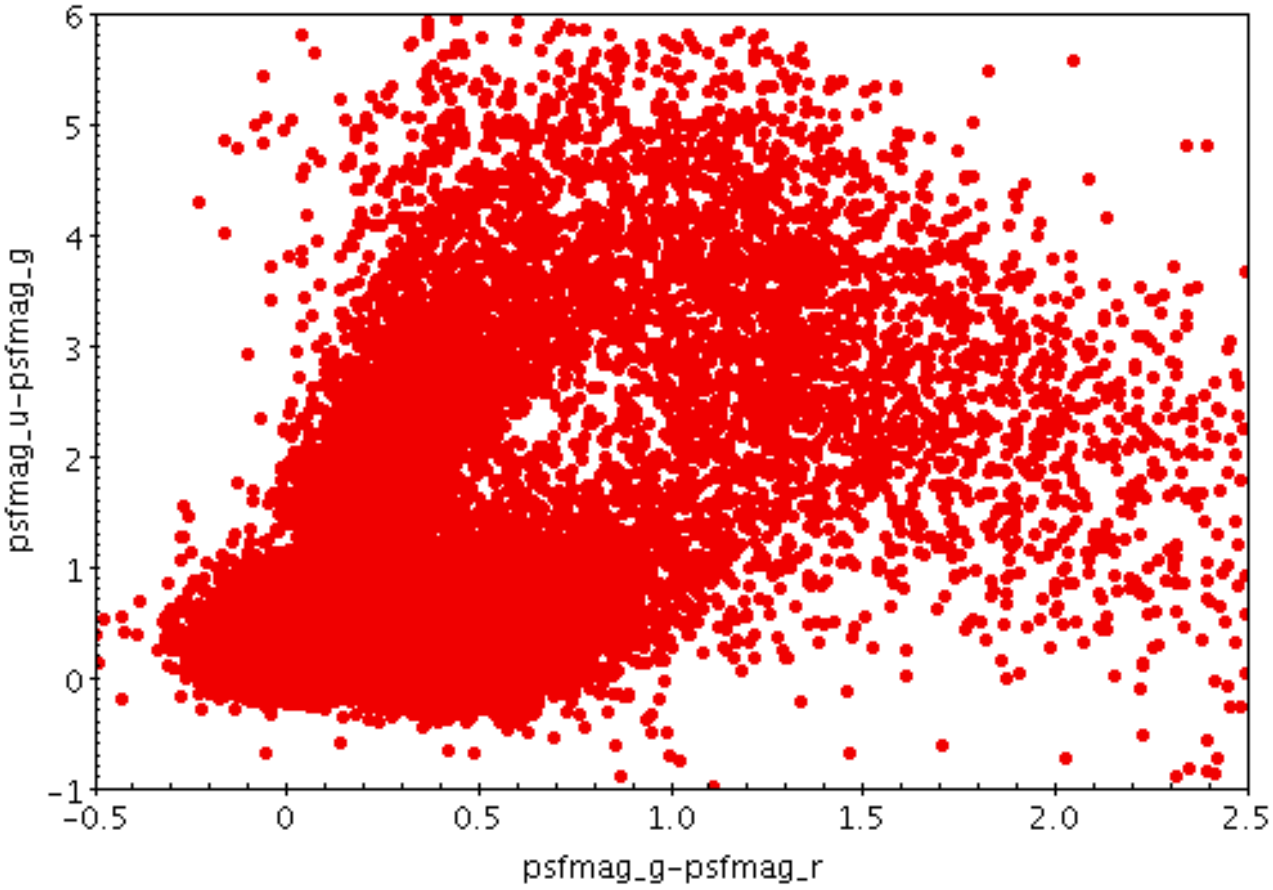
Paints markers in a single fixed colour.

Usage:

```
shadingN=flat colorN=<rrggbb>|red|blue|...
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane layer1=mark in1=dr5qso.fits
                  shading1=flat
                  x1=psfmag_g-psfmag_r y1=psfmag_u-psfmag_g size1=2
                  xmin=-0.5 xmax=2.5 ymin=-1 ymax=6
```

Associated parameters are as follows:

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to

list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

8.4.3 `translucent`

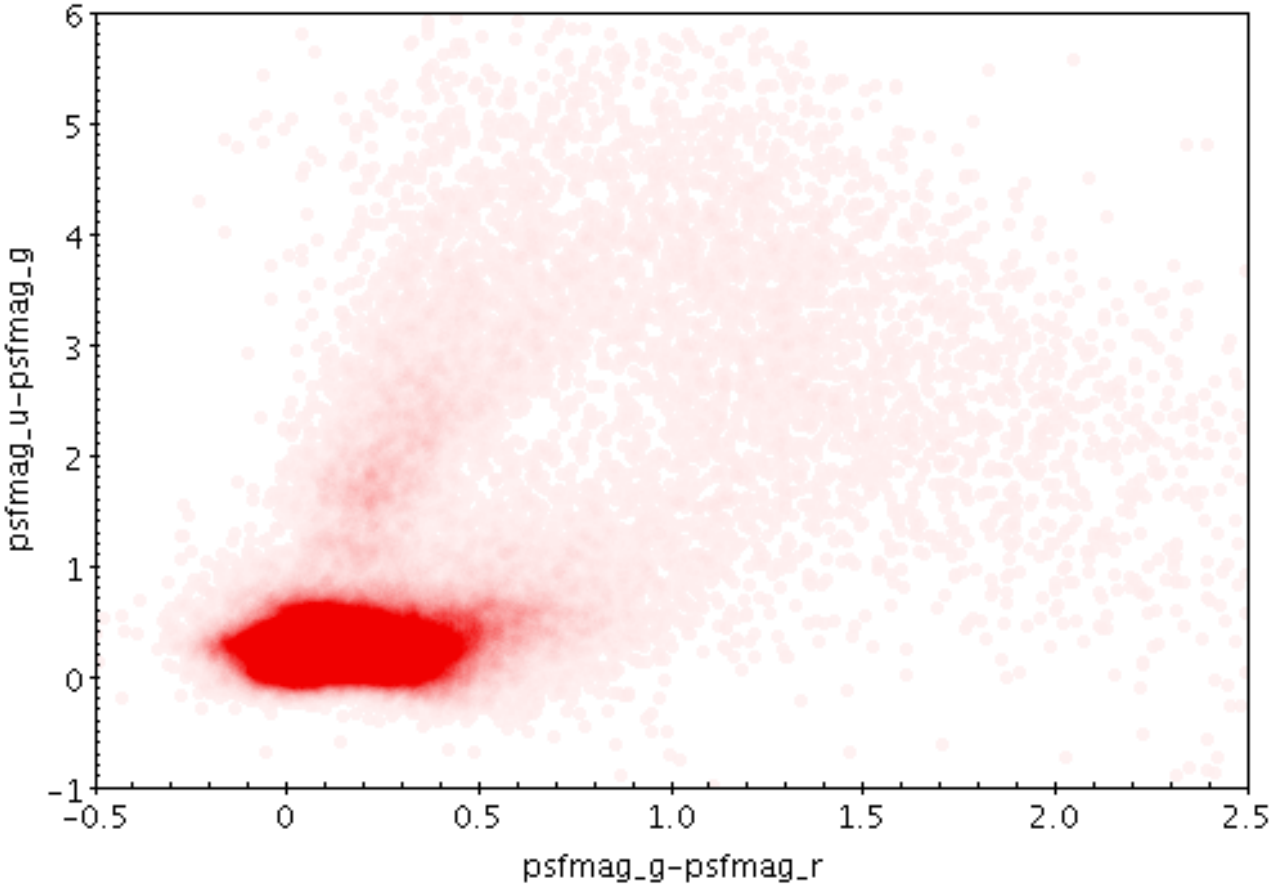
Paints markers in a transparent version of their selected colour. The degree of transparency is determined by how many points are plotted on top of each other and by the transparency level. Unlike transparent mode, the transparency varies according to the average point density in the plot, so leaving the setting the same as you zoom in and out usually has a sensible effect.

Usage:

```
shadingN=translucent colorN=<rrggbb>|red|blue|... translevelN=<number>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane layer1=mark in1=dr5qso.fits
                 shading1=translucent
                 x1=psfmag_g-psfmag_r y1=psfmag_u-psfmag_g size1=2
                 xmin=-0.5 xmax=2.5 ymin=-1 ymax=6
```

Associated parameters are as follows:

`colorN = <rrggbb>|red|blue|...` ***(Color)***
The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`translevelN = <number>` ***(Double)***
Sets the level of automatically controlled transparency. The higher this value the more transparent points are. Exactly how transparent points are depends on how many are currently being plotted on top of each other and the value of this parameter. The idea is that you can set it to some fixed value, and then get something which looks similarly transparent while you zoom in and out.

[Default: `0.1`]

8.4.4 `transparent`

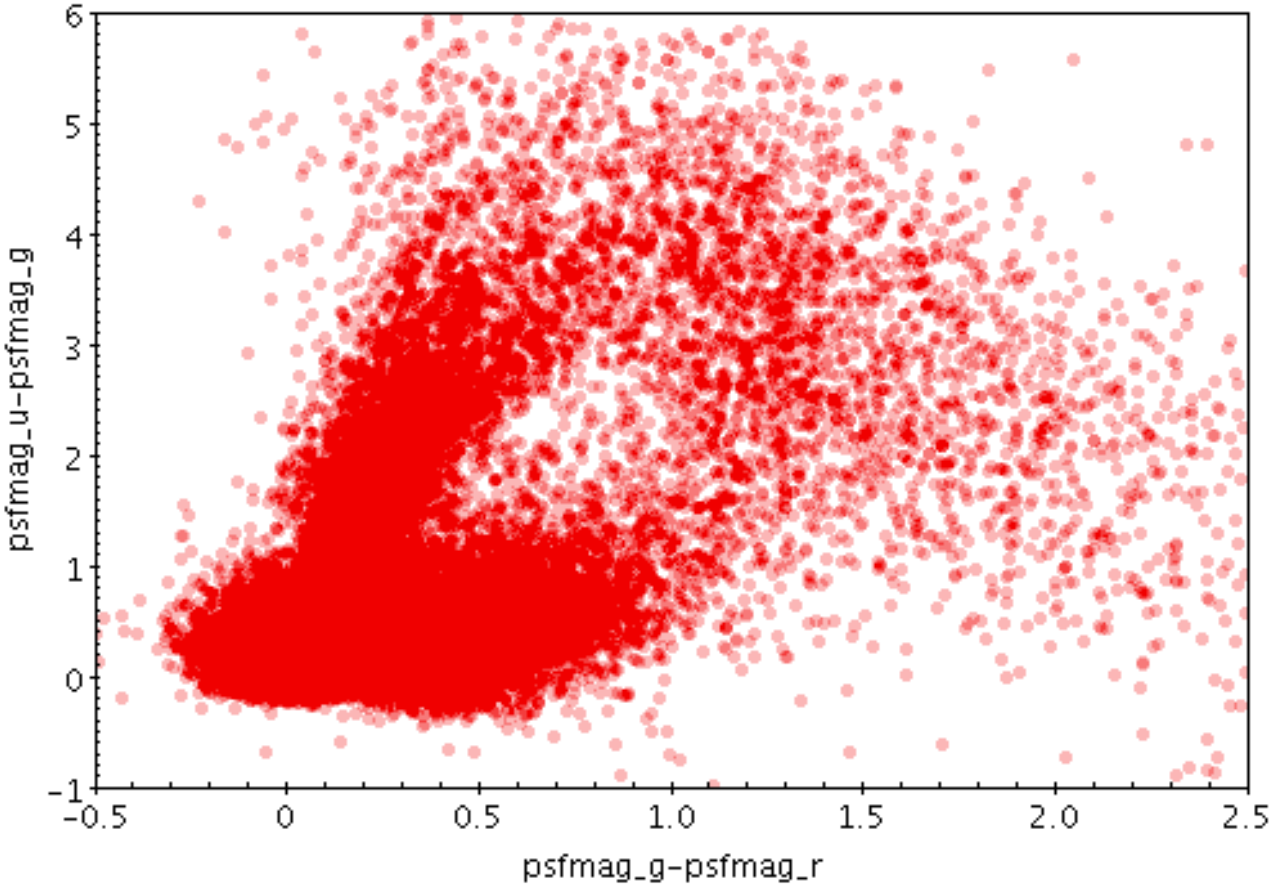
Paints markers in a transparent version of their selected colour. The degree of transparency is determined by how many points are plotted on top of each other and by the opaque limit. The opaque limit fixes how many points must be plotted on top of each other to completely obscure the background. This is set to a fixed value, so a transparent level that works well for a crowded region (or low magnification) may not work so well for a sparse region (or when zoomed in).

Usage:

```
shadingN=transparent colorN=<rrggbb>|red|blue|... opaqueN=<number>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane layer1=mark in1=dr5qso.fits
                  shading1=transparent
                  x1=psfmag_g-psfmag_r y1=psfmag_u-psfmag_g size1=2
                  xmin=-0.5 xmax=2.5 ymin=-1 ymax=6
```

Associated parameters are as follows:

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`opaqueN = <number>` ***(Double)***

The opacity of plotted points. The value is the number of points which have to be overplotted before the background is fully obscured.

[Default: `4`]

8.4.5 `density`

Paints markers using a configurable colour map to indicate how many points are plotted over each

other. Specifically, it colours each pixel according to how many times that pixel has has been covered by one of the markers plotted by the layer in question. To put it another way, it generates a false-colour density map with pixel granularity using a smoothing kernel of the form of the markers plotted by the layer. The upshot is that you can see the plot density of points or other markers plotted.

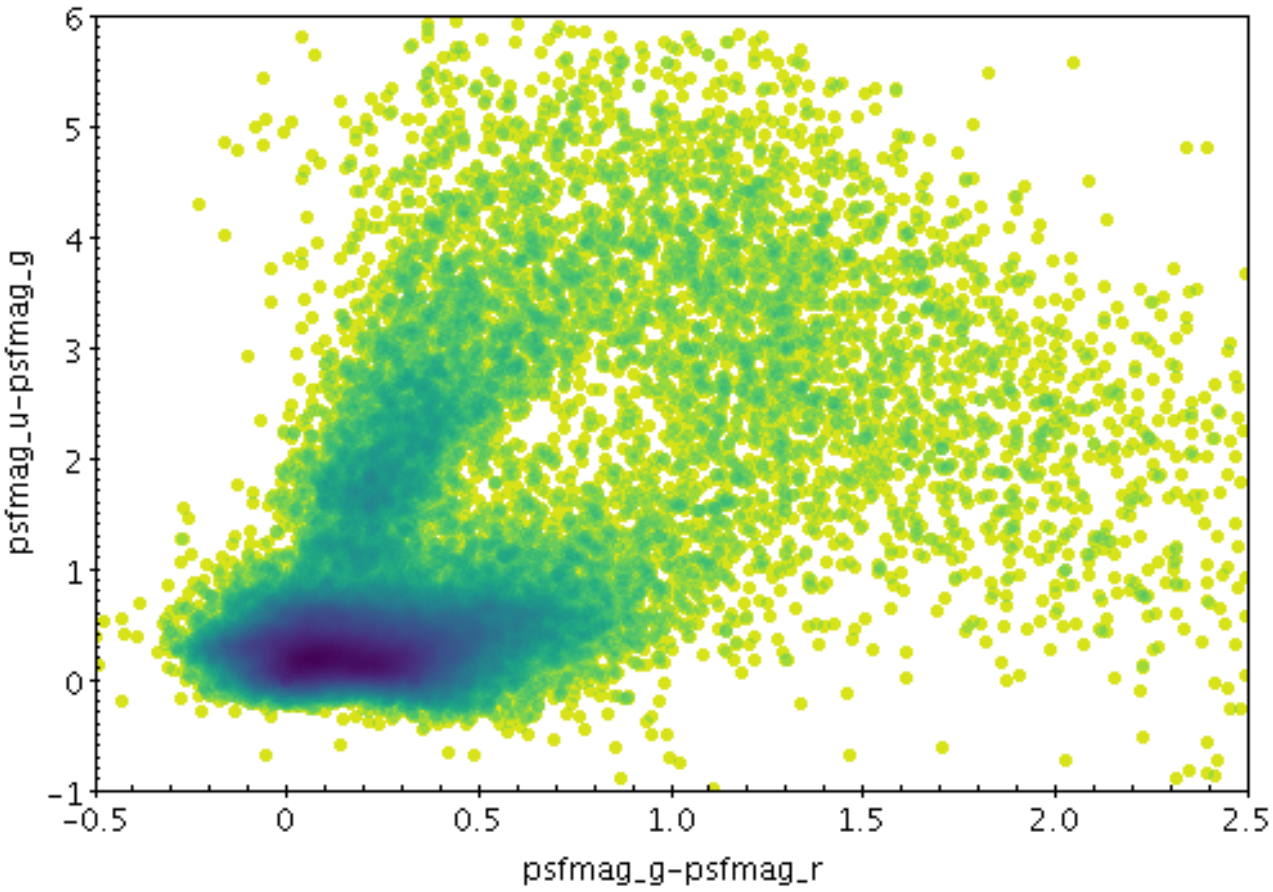
This is like auto mode, but with more user-configurable options.

Usage:

```
shadingN=density colorN=<rrggbb>|red|blue|...
                densemapN=<map-name>|<color>-<color>[-<color>...]
                denseclipN=<lo>,<hi> denseflipN=true|false
                densequantN=<number> densesubN=<lo>,<hi>
                densefuncN=log|linear|histogram|histolog|sqrt|square|acos|cos
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane layer1=mark in1=dr5qso.fits
                 shading1=density densemap1=viridis
                 x1=psfmag_g-psfmag_r y1=psfmag_u-psfmag_g size1=2
                 xmin=-0.5 xmax=2.5 ymin=-1 ymax=6
```

Associated parameters are as follows:

`colorN = <rrggbb>|red|blue|...` ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to

list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`denseclipN = <lo>,<hi>` ***(Subrange)***

Defines a subrange of the colour ramp to be used for Density shading. The value is specified as a (low,high) comma-separated pair of two numbers between 0 and 1.

If the full range `0,1` is used, the whole range of colours specified by the selected shader will be used. But if for instance a value of `0,0.5` is given, only those colours at the left hand end of the ramp will be seen.

If the null (default) value is chosen, a default clip will be used. This generally covers most or all of the range 0-1 but for colour maps which fade to white, a small proportion of the lower end may be excluded, to ensure that all the colours are visually distinguishable from a white background. This default is usually a good idea if the colour map is being used with something like a scatter plot, where markers are plotted against a white background. However, for something like a density map when the whole plotting area is tiled with colours from the map, it may be better to supply the whole range `0,1` explicitly.

`denseflipN = true|false` ***(Boolean)***

If true, the colour map on the Density axis will be reversed.

[Default: `false`]

`densefuncN = log|linear|histogram|histolog|sqrt|square|acos|cos` ***(Scaling)***

Defines the way that values in the Density range are mapped to the selected colour ramp.

The available options are:

- `log`: Logarithmic scaling
- `linear`: Linear scaling
- `histogram`: Scaling follows data distribution, with linear axis
- `histolog`: Scaling follows data distribution, with logarithmic axis
- `sqrt`: Square root scaling
- `square`: Square scaling
- `acos`: Arccos Scaling
- `cos`: Cos Scaling

For all these options, the full range of data values is used, and displayed on the colour bar if applicable (though it can be restricted using the `densesub` option) The `Linear`, `Log`, `Square` and `Sqrt` options just apply the named function to the full data range. The histogram options on the other hand use a scaling function that corresponds to the actual distribution of the data, so that there are about the same number of points (or pixels, or whatever is being scaled) of each colour. The histogram options are somewhat more expensive, but can be a good choice if you are exploring data whose distribution is unknown or not well-behaved over its min-max range. The `Histogram` and `HistoLog` options both assign the colours in the same way, but they display the colour ramp with linear or logarithmic annotation respectively; the `HistoLog` option also ignores non-positive values.

[Default: `log`]

`densemapN = <map-name>|<color>-<color>[-<color>...]` ***(Shader)***

Color map used for Density axis shading.

A mixed bag of colour ramps are available: `blacker`, `whiter`, `inferno`, `magma`, `plasma`,

`viridis`, `cividis`, `cubehelix`, `sron`, `rainbow`, `rainbow2`, `rainbow3`, `pastel`, `accent`, `gnuplot`, `gnuplot2`, `specxby`, `set1`, `paired`, `hotcold`, `rdbu`, `piyg`, `brbg`, `cyan-magenta`, `red-blue`, `brg`, `heat`, `cold`, `light`, `greyscale`, `colour`, `standard`, `bugn`, `bupu`, `orrd`, `pubu`, `purd`, `painbow`, `huecl`, `hue`, `intensity`, `rgb_red`, `rgb_green`, `rgb_blue`, `hsv_h`, `hsv_s`, `hsv_v`, `yuv_y`, `yuv_u`, `yuv_v`, `scale_hsv_s`, `scale_hsv_v`, `scale_yuv_y`. *Note:* many of these, including rainbow-like ones, are frowned upon by the visualisation community.

You can also construct your own custom colour map by giving a sequence of colour names separated by minus sign ("`-`") characters. In this case the ramp is a linear interpolation between each pair of colours named, using the same syntax as when specifying a colour value. So for instance "`yellow-hotpink-#0000ff`" would shade from yellow via hot pink to blue.

[Default: `blacker`]

`densequantN = <number>` ***(Double)***

Allows the colour map used for the Density axis to be quantised. If an integer value N is chosen then the colour map will be viewed as N discrete evenly-spaced levels, so that only N different colours will appear in the plot. This can be used to generate a contour-like effect, and may make it easier to trace the boundaries of regions of interest by eye.

If left blank, the colour map is nominally continuous (though in practice it may be quantised to a medium-sized number like 256).

`densesubN = <lo>,<hi>` ***(Subrange)***

Defines a normalised adjustment to the data range of the Density axis. The value may be specified as a comma-separated pair of two numbers, giving the lower and upper bounds of the range of of interest respectively. This sub-range is applied to the data range that would otherwise be used, either automatically calculated or explicitly supplied; zero corresponds to the lower bound and one to the upper.

The default value "`0,1`" therefore has no effect. The range could be restricted to its lower half with the value `0,0.5`.

[Default: `0,1`]

8.4.6 `aux`

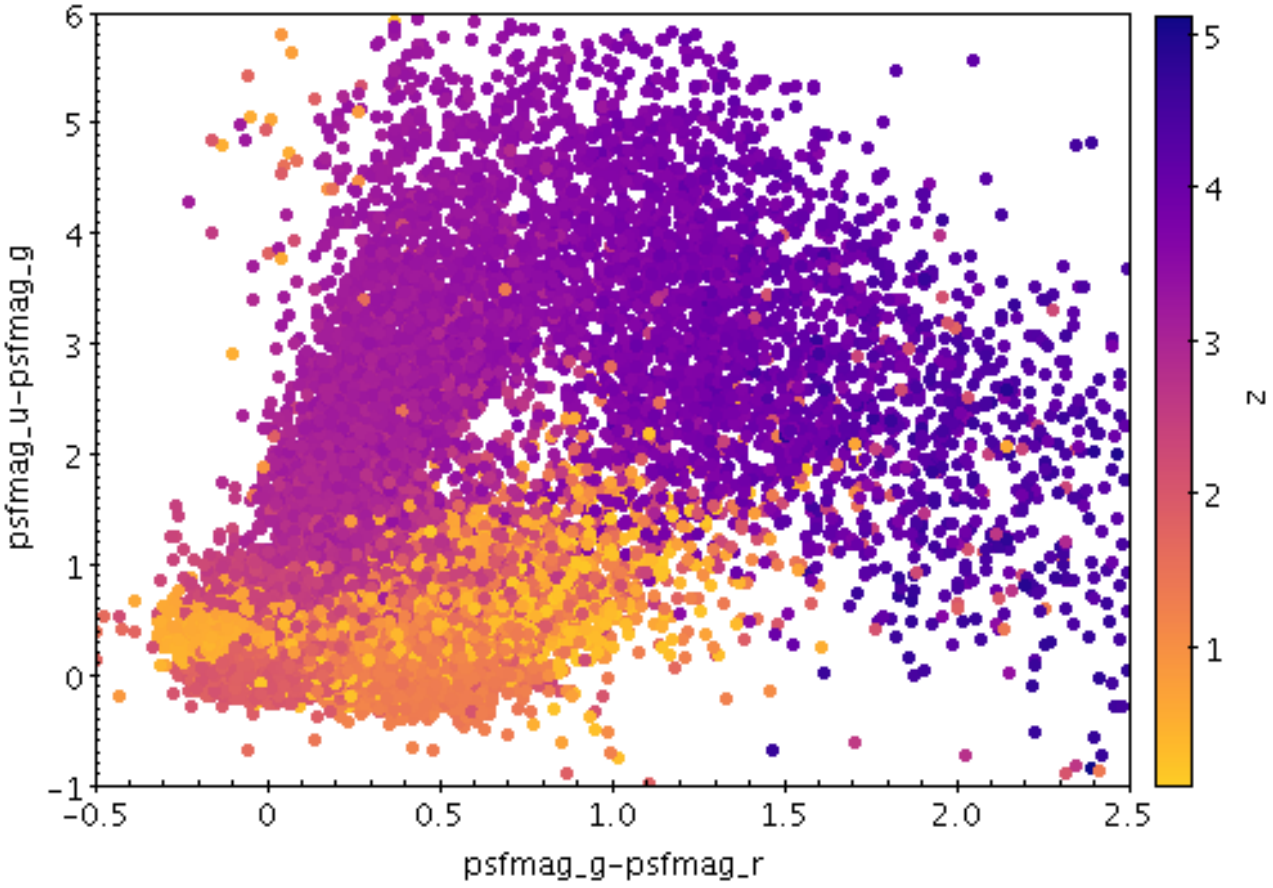
Paints markers in a colour determined by the value of an additional data coordinate. The marker colours then represent an additional dimension of the plot. You can also adjust the transparency of the colours used. The way that data values are mapped to colours is usually controlled by options at the level of the plot itself, rather than by per-layer configuration.

Usage:

```
shadingN=aux auxN=<num-expr> auxnullcolorN=<rrggbb>|red|blue|...
             opaqueN=<number>
```

All the parameters listed here affect only the relevant layer, identified by the suffix `N`.

Example:




```
stilts plot2plane layer1=mark in1=dr5qso.fits
                  shading1=aux aux1=z auxmap=plasma
                  x1=psfmag_g-psfmag_r y1=psfmag_u-psfmag_g size1=2
                  xmin=-0.5 xmax=2.5 ymin=-1 ymax=6
```

Associated parameters are as follows:

auxN = <num-expr> ***(String)***

Colour coordinate for Aux shading.

This parameter gives a column name, fixed value, or algebraic expression for the aux coordinate for layer N. The value is a numeric algebraic expression based on column names as described in Section 10.

auxnullcolorN = <rrggbb>|red|blue|... ***(Color)***

The color of points with a null value of the Aux coordinate, given by name or as a hexadecimal RGB value.

The standard plotting colour names are red, blue, green, grey, magenta, cyan, orange, pink, yellow, black, light_grey, white. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from AliceBlue to YellowGreen, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "#" or "0x", giving red, green and blue intensities, e.g. "ff00ff", "#ff00ff" or "0xff00ff" for magenta.

If the value is null, then points with a null Aux value will not be plotted at all.

[Default: grey]

opaqueN = <number> ***(Double)***

The opacity of points plotted in the Aux colour. The value is the number of points which have

to be overplotted before the background is fully obscured.

[Default: 1]

8.4.7 weighted

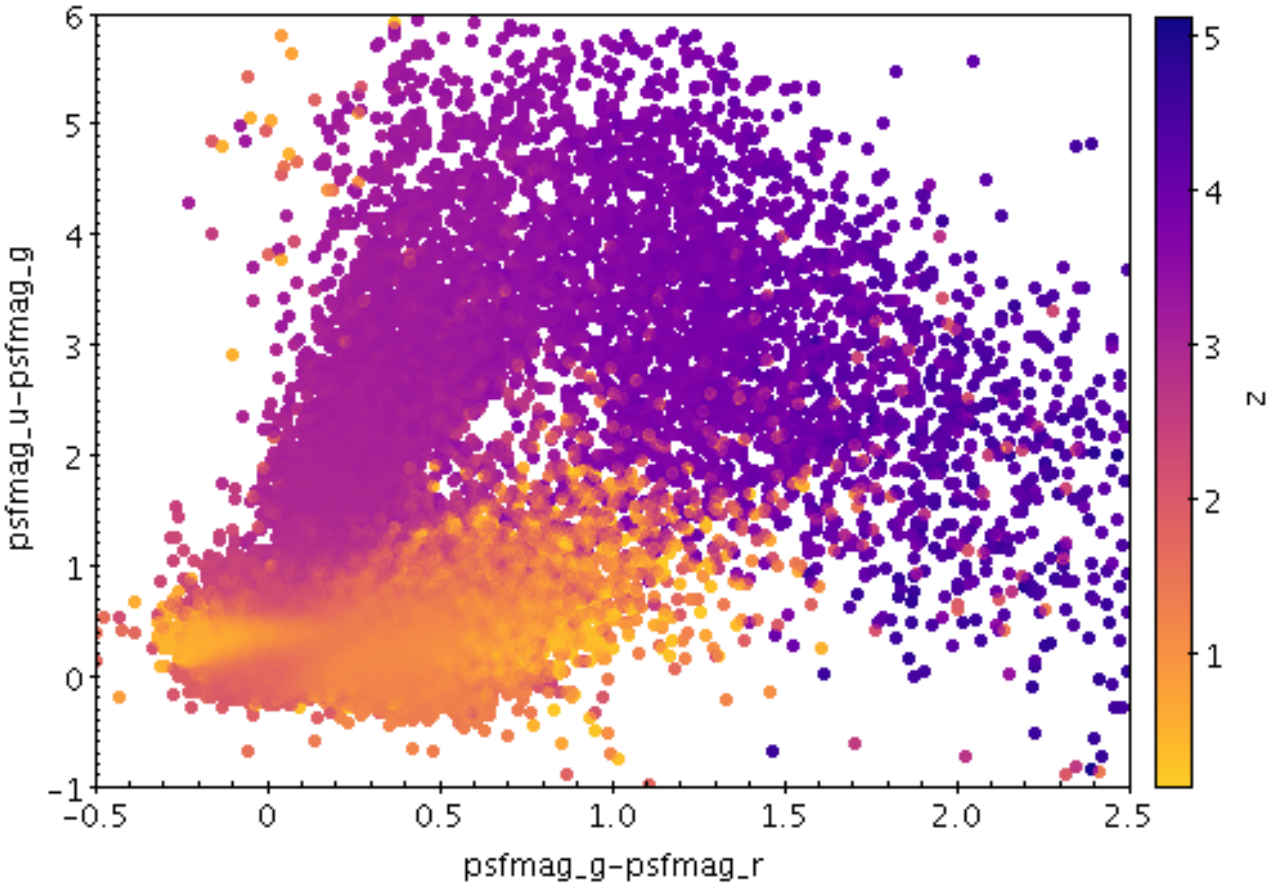
Paints markers like the Density mode, but with optional weighting by an additional coordinate. You can configure how the weighted coordinates are combined to give the final weighted result. The way that data values are mapped to colours is usually controlled by options at the level of the plot itself, rather than by per-layer configuration.

Usage:

```
shadingN=weighted weightN=<num-expr> colorN=<rrggbb>|red|blue|...
                  combineN=sum|count|mean|median|q1|q3|min|max|stdev|hit
```

All the parameters listed here affect only the relevant layer, identified by the suffix N.

Example:




```
stilts plot2plane layer1=mark in1=dr5qso.fits
                  shading1=weighted weight1=z auxmap=plasma
                  x1=psfmag_g-psfmag_r y1=psfmag_u-psfmag_g size1=2
                  xmin=-0.5 xmax=2.5 ymin=-1 ymax=6
```

Associated parameters are as follows:

colorN = <rrggbb>|red|blue|... ***(Color)***

The color of plotted data, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `red`]

`combineN = sum|count|mean|median|q1|q3|min|max|stdev|hit` ***(Combiner)***

Defines how values contributing to the same pixel are combined together to produce the value assigned to that pixel (and hence its colour).

When a weight is in use, `mean` or `sum` are typically sensible choices. If there is no weight (a pure density map) then `count` is usually better, but in that case it may make more sense (it is more efficient) to use one of the other shading modes instead.

The available options are:

- `sum`: the sum of all the combined values per bin
- `count`: the number of non-blank values per bin (weight is ignored)
- `mean`: the mean of the combined values
- `median`: the median
- `q1`: first quartile
- `q3`: third quartile
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `stdev`: the sample standard deviation of the combined values
- `hit`: 1 if any values present, NaN otherwise (weight is ignored)

[Default: `mean`]

`weightN = <num-expr>` ***(String)***

Weight coordinate for weighted density shading.

This parameter gives a column name, fixed value, or algebraic expression for the `weight` coordinate for layer `N`. The value is a numeric algebraic expression based on column names as described in Section 10.

8.5 Output Modes

The plots generated by the plotting commands can be used in various different ways. One thing you might want to do is to write the output to a file in a given graphics format (`out`); another is to preview it directly on the screen (`swing`). By default one or other of these will happen depending on whether you specify an output file. However there are other possibilities; these are listed in the following subsections.

Except for display to the screen, these modes should work happily on a headless machine (one with no graphics display, as may be the case for a web server). When running headless, you may find it necessary to set the java system property "`java.awt.headless`" to `true` - see Section 3.3.

The default output mode is `auto`, which means that output is to a file if an output file is specified, or to the screen if it is not. So in most cases you don't need to specify the `omode` parameter explicitly.

8.5.1 `swing`

Usage:

```
omode=swing
```

Plot will be displayed in a window on the screen. This plot is "live"; it can be resized and (except for old-style plots) navigated around with mouse actions in the same way as plots in TOPCAT.

8.5.2 `out`

Usage:

```
omode=out out=<out-file> ofmt=png|png-transp|gif|jpeg|pdf|svg|eps|eps-gzip
```

Plot will be written to a file given by `out` using the graphics format given by `ofmt`.

8.5.3 `cgi`

Usage:

```
omode=cgi ofmt=png|png-transp|gif|jpeg|pdf|svg|eps|eps-gzip
```

Plot will be written in a way suitable for CGI use direct from a web server. The output is in the graphics format given by `ofmt`, preceded by a suitable "Content-type" declaration.

8.5.4 `discard`

Usage:

```
omode=discard
```

Plot is drawn, but discarded. There is no output.

8.5.5 `auto`

Usage:

```
omode=auto [out=<out-file>]
```

Behaves as `swing` or `out` mode depending on presence of `out` parameter

8.6 Export Formats

Several of the plot output modes write the plot in some graphics format or other. When selecting an output format it is important to understand the distinction between *bitmapped* and *vector* formats; basically bitmapped formats represent the image as a grid of finite-sized pixels while vector formats notionally draw smooth lines. Bitmapped formats are fine for a computer screen, but for high quality paper printouts you will want a vector format. You can convert from vector to bitmapped but not (usefully) in the other direction. There are a couple of subtleties to this distinction specific to STILTS graphical output as discussed below.

The following formats are the available values for the `ofmt` parameter of the various plot commands:

`png`
: PNG format. This is a flexible bitmapped format providing transparency and an unlimited number of colours with good lossless compression. It is widely supported by non-ancient

browsers and other image viewers, and is generally recommended for bitmapped output.

gif
GIF format. This is a bitmapped format providing transparency and lossless compression. The number of colours is limited to 255 however, so if you are using auxiliary axes (colour variation to represent higher dimensionality) or other plot features which use a wide range of colours you may see image degradation. It has long been widely supported by browsers and other image viewers.

jpeg
JPEG format. This is a bitmapped format with lossy compression intended primarily for photographs. Transparency is not supported, and although there is no limit on the maximum number of colours, its lossiness means that plots generated using it generally look a bit smudged.

pdf
Portable Document Format. This is the format which can be read by Adobe's Acrobat Reader. It is a widely portable vector format, and is suitable for printing at high resolution, either standalone or imported into some other presentation format. However, there are a couple of caveats when it comes to using it with STILTS plots.

1. If used to plot a very large number of points, the output PDF file can get quite large, though it's much better than for `eps` output (see below).
2. For certain colour shading options (auto, density, and in some circumstances transparency), the body of the plot will be drawn as a bitmap rather than vector graphics. This is sometimes a blessing in disguise since with very large numbers of points a vector PDF file could get unmanageably large in any case. In this case the interior of the plot will be pixellated. The axes and annotations outside of the plot will still be drawn in vector format however.

svg
Scalable Vector Graphics. This is an XML-based vector graphics format developed for display in web pages, and defined by the W3C. This exporter can generate OutOfMemoryErrors if asked to generate a large output file.

eps
Encapsulated Postscript. This is a vector format which is suitable for printing at high resolution either standalone or imported into some other presentation format (you may need to convert it via PDF depending on the intended destination). However, there are a couple of caveats when it comes to using it with STILTS plots.

1. Unfortunately the postscript driver used by STILTS is not very efficient and can result in large, sometimes very large, postscript output files. This is likely to be a problem for plots with a large number of non-transparent points. For this reason `eps-gzip` or `pdf` may be a better choice.
2. Postscript has no support for partial transparency, so if plots are drawn with partially transparent points (common for very large data sets) the only way they can be rendered is by drawing the body of the plot as a bitmap rather than as vector graphics. This is sometimes a blessing in disguise since with very large numbers of points a vector postscript file would likely be unmanageably large in any case. So if there is any transparency in the plot, the interior of the plot will be pixellated. The axes and annotations outside of the plot will still be drawn in vector format however.

eps-gzip
Just like the `eps` format above except that the output is automatically compressed using the GZIP format as it is written. Postscript compresses well (typically a factor of 5-10).

9 Old-Style Plotting

This section describes deprecated commands. For recommended plotting commands, see Section 8.

From version 2.0 (October 2008), STILTS incorporated three table plotting commands:

- `plot2d`: Old-style 2D Scatter Plot
- `plot3d`: Old-style 3D Scatter Plot
- `plothist`: Old-style Histogram

These provided command-line access to some, though not all, of the plotting capabilities offered by TOPCAT.

Since version 3.0 (October 2014), these commands are deprecated in favour of the more powerful ones described in Section 8. The rest of this section describes some aspects of the deprecated commands for the benefit of legacy code. The output modes and formats are the same in old- and new-style plots, and are discussed in Section 8.5 and Section 8.6. The handling of parameters and suffixes for these commands is not quite the same as for new-style plots, and is documented in the next subsection.

As a simple example, if a file "cat.fits" contains the columns RMAG and BMAG for red and blue magnitudes, you can draw a two-dimensional colour-magnitude scatter plot with the command:

```
stilts plot2d in=cat.fits xdata=BMAG-RMAG ydata=BMAG
```

Since an output file is not specified, the plot is shown on the screen for convenience. To send the output to a PNG file, do instead:

```
stilts plot2d in=cat.fits xdata=BMAG-RMAG ydata=BMAG out=plot.png ofmt=png
```

in some cases (including the above), the `ofmt` parameter is not required since STILTS may be able to guess the format from the output file name. Various other options for output and graphics formats are described in Section 8.5 and Section 8.6

Some of the parameters use suffixes to define data sets and therefore behave a bit differently from the parameters elsewhere in STILTS - a discussion of these is given in the following subsection. Some other plotting-specific topics are also discussed below.

9.1 Parameter Suffixes

This section describes deprecated commands. For recommended plotting commands, see Section 8.

Some of the parameters for the plotting tasks behave a little bit differently to other parameters in STILTS, in order to accommodate related sets of values. If you look at the usage of one of the plotting commands, for instance in Appendix B.15.1, you will see that a number of the parameters have the suffixes "`N`" or "`NS`". These suffixes can be substituted with any convenient string to identify parameters which relate to the same input datasets or subsets. Specifically:

Suffix "`N`":
Denotes an input dataset. At least the `inN` parameter must be given to identify the source of the data; any other parameters with the same value of the `N` suffix relate to that dataset. A *dataset* here refers to a particular set of plot data from a table; in most cases each input table corresponds to a different dataset, though two datasets may correspond to different sets of columns from the same table.

Suffix "`NS`":
Denotes a particular subset of the rows in dataset `N`. At least the `subsetNS` parameter must be given to identify the expression by which the subset is defined; any other parameters with the

same value of the NS suffix relate to that subset.

Some examples will help to illustrate. The following will generate a Cartesian plot of catalogue position from a single dataset:

```
stilts plot2d in=gals.fits xdata=RA ydata=DEC
```

In this case the N suffix is present on each of the parameters in, xdata and ydata, but is equal to the empty string, hence invisible. This is perfectly legal, and convenient when only a single table is in use. If we wish to overplot two datasets however, the dataset suffixes (or one of them at least) have to be made explicit so that different ones can be used, for instance:

```
stilts plot2d in1=gals.fits  xdata1=RA      ydata1=DEC
              in2=stars.fits xdata2=RAJ2000 ydata2=DEJ2000
```

The suffix values "1" and "2" are quite arbitrary and can be chosen as convenient, so the following would do exactly the same as the previous example:

```
stilts plot2d in_GAL=gals.fits   xdata_GAL=RA       ydata_GAL=DEC
              in_STAR=stars.fits xdata_STAR=RAJ2000 ydata_STAR=DEJ2000
```

The other parameters which have the N suffix apply only to the matching dataset, so for instance the following:

```
stilts plot2d in1=gals.fits  xdata1=RA      ydata1=DEC     txtlabel1=NGC_ID
              in2=stars.fits xdata2=RAJ2000 ydata2=DEJ2000
```

would draw text labels adjacent to the points from only the gals.fits file giving the contents of its NGC_ID column.

The NS suffix identifies distinct *row subsets* within the same or different datasets. A subset is defined by supplying a boolean inclusion expression (each row is included only if the expression evaluates true for that row) as the value of a subsetNS parameter. If, as in all the examples we have seen so far, no subsetNS parameter is supplied for a given dataset, then it is treated as a special case, as if a single subset with a name equal to the empty string (S="") containing all rows has been specified. So our earlier simple example:

```
stilts plot2d in=gals.fits xdata=RA ydata=DEC
```

is equivalent to

```
stilts plot2d in=gals.fits xdata=RA ydata=DEC subset=true
```

If we wish to split the plotted points into two sets based on their R-B colours, we can write something like:

```
stilts plot2d in=gals.fits xdata=RA ydata=DEC
              subsetX='RMAG-BMAG>0' subsetY='RMAG-BMAG<=0'
```

This will generate a plot with two subsets shown using different colours and/or plotting symbols. These colours and symbols are selected automatically. More control over the appearance can be exercised by setting values for some of the other parameters with NS suffixes, for instance

```
stilts plot2d in=gals.fits xdata=RA ydata=DEC
                           subset_A='RMAG-BMAG>0'  colour_A=blue
                           subset_B='RMAG-BMAG<=0' colour_B=red
```

Again, the suffix strings can be chosen to have any value as convenient.

The dataset- and subset-specific parameters must be put together if there are multiple datasets with multiple subsets to plot simultaneously, for instance:

```
stilts plot2d in_1=gals.fits  xdata_1=RA ydata_1=DEC
                              subset_1_A='RMAG-BMAG>0'  colour_1_A=blue
                              subset_1_B='RMAG-BMAG<=0' colour_1_B=red
              in_2=stars.fits xdata_2=RAJ2000 ydata_2=DEJ2000
                              colour_2=green
```

Finally, it's not quite true that the suffixes chosen have no effect on the plot; they may influence the order in which sets are plotted. Markers drawn for sets plotted earlier may be obscured by the markers drawn for sets plotted later, so this can affect the appearance of the plot. If you want to control this, use the `sequence` parameter. For instance, to ensure that star data appears on top of galaxy data in the plot, do the following:

```
stilts plot2d in_GAL=gals.fits   xdata_GAL=RA       ydata_GAL=DEC
              in_STAR=stars.fits xdata_STAR=RAJ2000 ydata_STAR=DEJ2000
              sequence=_GAL,_STAR
```

More examples can be found in the **Examples** subsections of the individual plotting command descriptions in Appendix B.

10 Algebraic Expression Syntax

Many of the STILTS commands allow you to use algebraic expressions based on table columns when doing things like making row selections, defining new columns, selecting values to plot or match, and so on. In these cases you are defining an expression which has a value in each row as a function of the values in the existing columns in that row. This is a powerful feature which permits you to manipulate and select table data in very flexible ways. The syntax for entering these expressions is explained in this section.

What you write are actually expressions in the Java language, which are compiled into Java bytecode before evaluation. However, this does not mean that you need to be a Java programmer to write them. The syntax is pretty similar to C, but even if you've never programmed in C most simple things, and many complicated ones, are quite intutitive.

The following explanation gives some guidance and examples for writing these expressions. Unfortunately a complete tutorial on writing Java expressions is beyond the scope of this document, but it should provide enough information for even a novice to write useful expressions.

The expressions that you can write are basically any function of all the column values which apply to a given row; the function result can then be used where STILTS needs a per-row value, for instance to define a new column. If the built-in operators and functions are not sufficient, or it's unwieldy to express your function in one line of code, it is possible to add new functions by writing your own classes - see Section 10.8.3.

Note that since these algebraic expressions often contain spaces, you may need to enclose them in single or double quotes so that they don't get confused with other parts of the command string.

Note: if Java is running in an environment with certain security restrictions (a security manager which does not permit creation of custom class loaders) then algebraic expressions won't work at all. It's not particularly likely that security restrictions will be in place if you are running from the command line though.

10.1 Referencing Column Values

To create a useful expression which can be evaluated for each row in a table, you will have to refer to cells in different columns of that row. You can do this in several ways:

By Name
: The Name of the column may be used if it is unique (no other column in the table has the same name) and if it has a suitable form. This means that it must have the form of a Java variable - basically starting with a letter and continuing with letters, numbers, underscores and currency symbols. In particular it cannot contain spaces, commas, parentheses etc.

  As a special case, if an expression contains just a single column name, rather than some more complicated expression, then any column name may be used, even one containing non-alphanumeric characters.

  Column names are treated case-insensitively.

By $ID
: The "$ID" identifier of the column may always be used to refer to it; this is a useful fallback if the column name isn't suitable for some reason (for instance it contains spaces or is not unique). This is just a "$" sign followed by the column index - the first column is $1.

By ucd$ specifier
: If the column has a Unified Content Descriptor (this will usually only be the case for VOTable or possibly FITS format tables) you can refer to it using an identifier of the form

ucd$<ucd-spec>". Depending on the version of UCD scheme used, UCDs can contain various punctuation marks such as underscores, semicolons and dots; for the purpose of this syntax these should all be represented as underscores ("_"). So to identify a column which has the UCD "phot.mag;em.opt.R", you should use the identifier "ucd$phot_mag_em_opt_r". Matching is not case-sensitive. Futhermore, a trailing underscore acts as a wildcard, so that the above column could also be referenced using the identifier "ucd$phot_mag_". If multiple columns have UCDs which match the given identifer, the first one will be used.

Note that the same syntax can be used for referencing table parameters (see the next section); columns take preference so if a column and a parameter both match the requested UCD, the column value will be used.

By utype$ specifier

If the column has a **Utype** (this will usually only be the case for VOTable or possibly FITS format tables) you can refer to it using an identifier of the form "utype$<utype-spec>". Utypes can contain various punctuation marks such as colons and dots; for the purpose of this syntax these should all be represented as underscores ("_"). So to identify a column which has the Utype "ssa:Access.Format", you should use the identifier "utype$ssa_Access_Format". Matching is not case-sensitive. If multiple columns have Utypes which match the given identifier, the first one will be used.

Note that the same syntax can be used for referencing table parameters (see the next section); columns take preference so if a column and a parameter both match the requested Utype, the column value will be used.

Using value*() functions

You can use the special functions valueDouble, valueInt, valueLong, valueString and valueObject to obtain the typed value of a column with a given name. The argument of the function is a string giving the exact (case-sensitive) column name, for instance valueDouble("b_E(BP-RP)") will yield the value of the column named "b_E(BP-RP)" as a double-precision floating point value. These functions are *not* the generally recommended way to get column values, since they are slower and provide less type-checking than the other options listed above, and can occasionally lead to some other esoteric problems. However, if you need to refer by name to strangely-named columns they are sometimes a convenient option.

With the Object$ prefix

If a column is referenced with the prefix "Object$" before its identifier (e.g. "Object$BMAG" for a column named BMAG) the result will be the column value as a java Object. Without that prefix, numeric columns are evaluated as java primitives. In most cases, you *don't want to do this*, since it means that you can't use the value in arithmetic expressions. However, if you need the value to be passed to a (possibly user-defined) method, and you need that method to be invoked even when the value is null, you have to do it like this. Null-valued primitives otherwise cause expression evaluation to abort.

The value of the variables so referenced will be a primitive (boolean, byte, short, char, int, long, float, double) if the column contains one of the corresponding types. Otherwise it will be an Object of the type held by the column, for instance a String. In practice this means: you can write the name of a column, and it will evaluate to the numeric (or string) value that that column contains in each row. You can then use this in normal algebraic expressions such as "B_MAG-U_MAG" as you'd expect.

10.2 Referencing Parameter Values

Some tables have constant values associated with them; these may represent such things as the epoch at which observations were taken, the name of the catalogue, an angular resolution associated with all observations, or any number of other things. Such constants are known as *table parameters* (not to be confused with parameters passed to STILTS commands) and can be thought of as extra

columns which have the same value for every row. The values of such parameters can be referenced in STILTS algebraic expressions as follows:

param$*name*
If the parameter name has a suitable form (starting with a letter and continuing with letters or numbers) it can be referenced by prefixing that name with the string `param$`.

ucd$*ucd-spec*
If the parameter has a Unified Content Descriptor it can be referenced by prefixing the UCD specifier with the string `ucd$`. Any punctuation marks in the UCD should be replaced by underscores, and a trailing underscore is interpreted as a wildcard. See Section 10.1 for more discussion.

utype$*utype-spec*
If the parameter has a Utype, it can be referenced by prefixing the Utype specifier with the string `utype$`. Any punctuation marks in the Utype should be replaced by underscores. See Section 10.1 for more discussion.

Note that if a parameter has a name in an unsuitable form (e.g. containing spaces) and has no UCD then it cannot be referenced in an expression. One possible workaround for that is to use the `fixcolnames` filter.

10.3 Special Tokens

There are a few pseudo-variables which have special functions in the expression language. The following specials are column-like, in that they have a different value for each row:

$index *or* $0
The value of this is the current row number (the first row is 1). Note that this value is a `long` (8-byte integer); when using it in certain expressions you may find it necessary to convert it to an `int` (4-byte integer) using the `toInteger()` function. *The deprecated alias `"INDEX"` may also be used.*

$random *(Deprecated)*
Evaluates to a double-precision random number `0<=x<1`. **NOTE:** this token is deprecated since it can behave unpredictably (the same cell does not always yield the same result). Use instead the `random()` function in class Maths (Section 10.6.14).

The following specials are parameter-like, in that their value is not sensitive to the row:

$ncol
The number of columns in the table.

$nrow
The number of rows in the table. Note in some cases this is not known (e.g. if the table is being streamed), in which case the value of this variable is null. Note also that this value is a `long` (8-byte integer); when using it in certain expressions you may find it necessary to convert it to an `int` (4-byte integer) using the `toInteger()` function.

10.4 Null Values

When no special steps are taken, if a null value (blank cell) is encountered in evaluating an expression (usually because one of the columns it relies on has a null value in the row in question) then the result of the expression is also null.

It is possible to exercise more control than this, but it requires a little bit of care, because the expressions work in terms of primitive values (numeric or boolean ones) which don't in general

have a defined null value. The name "`null`" in expressions gives you the java `null` reference, but this cannot be matched against a primitive value or used as the return value of a primitive expression.

For most purposes, the following two tips should enable you to work with null values:

Testing for null
To test whether a column contains a null value, prepend the string "`NULL_`" (use upper case) to the column name or $ID. This will yield a boolean value which is true if the column contains a blank, and false otherwise.

Returning null
To return a null value from a numeric expression, use the name "`NULL`" (upper case). To return a null value from a non-numeric expression (e.g. a String column) use the name "`null`" (lower case).

Null values are often used in conjunction with the conditional operator, "`? :`"; the expression

```
test ? tval : fval
```

returns the value `tval` if the boolean expression `test` evaluates true, or `fval` if `test` evaluates false. So for instance the following expression:

```
Vmag == -99 ? NULL : Vmag
```

can be used to define a new column which has the same value as the `Vmag` column for most values, but if `Vmag` has the "magic" value -99 the new column will contain a blank. The opposite trick (substituting a blank value with a magic one) can be done like this:

```
NULL_Vmag ? -99 : Vmag
```

Some more examples are given in Section 10.7.

Note that for floating point data, STILTS treats `null` and NaN (Not-a-Number) values somewhat interchangeably. Blank values arising either from an input file format that can represent missing values, or from processing that fails to provide a definite value, are in most cases represented internally as `null` for integer-type values and NaN for floating point values. However in general users should not rely on distinguishing between `null` and NaN.

10.5 Operators

The operators are pretty much the same as in the C language. The common ones are:

Arithmetic

`+` (add)
`-` (subtract)
`*` (multiply)
`/` (divide)
`%` (modulus)

Boolean

`!` (not)
`&&` (and)
`||` (or)
`^` (exclusive-or)
`==` (numeric identity)
`!=` (numeric non-identity)

< (less than)
> (greater than)
<= (less than or equal)
>= (greater than or equal)

Bitwise

& (and)
| (or)
^ (exclusive-or)
<< (left shift)
>> (right shift)
>>> (logical right shift)

Numeric Typecasts

(byte) (numeric -> signed byte)
(short) (numeric -> 2-byte integer)
(int) (numeric -> 4-byte integer)
(long) (numeric -> 8-byte integer)
(float) (numeric -> 4-type floating point)
(double) (numeric -> 8-byte floating point)

Note you may find the Maths (Section 10.6.14) conversion functions more convenient for numeric conversions than these.

Other

+ (string concatenation)
[] (array dereferencing - first element is zero)
?: (conditional switch)
instanceof (class membership)

10.6 Functions

Many functions are available for use within your expressions, covering standard mathematical and trigonometric functions, arithmetic utility functions, type conversions, and some more specialised astronomical ones. You can use them in just the way you'd expect, by using the function name (unlike column names, this is case-sensitive) followed by comma-separated arguments in brackets, so

```
max(IMAG,JMAG)
```

will give you the larger of the values in the columns IMAG and JMAG, and so on.

The functions available for use by default are listed by class in the following subsections with their arguments and short descriptions. The `funcs` command provides another way to browse these function descriptions online.

10.6.1 Arithmetic

Standard arithmetic functions including things like rounding, sign manipulation, and maximum/minimum functions. Phase folding operations, and a convenient form of the modulus operation on which they are based, are also provided.

`roundUp( x )`
Rounds a value up to an integer value. Formally, returns the smallest (closest to negative

infinity) integer value that is not less than the argument.

- Parameters:
  - x *(floating point)*: a value.
- Return value
  - *(integer)*: x rounded up

`roundDown( x )`
Rounds a value down to an integer value. Formally, returns the largest (closest to positive infinity) integer value that is not greater than the argument.

- Parameters:
  - x *(floating point)*: a value
- Return value
  - *(integer)*: x rounded down

`round( x )`
Rounds a value to the nearest integer. Formally, returns the integer that is closest in value to the argument. If two integers are equally close, the result is the even one.

- Parameters:
  - x *(floating point)*: a floating point value.
- Return value
  - *(integer)*: x rounded to the nearest integer

`roundDecimal( x, dp )`
Rounds a value to a given number of decimal places. The result is a `float` (32-bit floating point value), so this is only suitable for relatively low-precision values. It's intended for truncating the number of apparent significant figures represented by a value which you know has been obtained by combining other values of limited precision. For more control, see the functions in the `Formats` class.

- Parameters:
  - x *(floating point)*: a floating point value
  - `dp` *(integer)*: number of decimal places (digits after the decimal point) to retain
- Return value
  - *(floating point)*: floating point value close to x but with a limited apparent precision
- Example:
  - `roundDecimal(PI,2) = 3.14f`

`abs( x )`
Returns the absolute value of an integer value. If the argument is not negative, the argument is returned. If the argument is negative, the negation of the argument is returned.

- Parameters:
  - x *(integer)*: the argument whose absolute value is to be determined
- Return value

- *(integer)*: the absolute value of the argument.

`abs( x )`
Returns the absolute value of a floating point value. If the argument is not negative, the argument is returned. If the argument is negative, the negation of the argument is returned.

- Parameters:
  - `x` *(floating point)*: the argument whose absolute value is to be determined
- Return value
  - *(floating point)*: the absolute value of the argument.

`max( a, b )`
Returns the greater of two integer values. If the arguments have the same value, the result is that same value.

Multiple-argument maximum functions are also provided in the `Arrays` and `Lists` packages.

- Parameters:
  - `a` *(integer)*: an argument.
  - `b` *(integer)*: another argument.
- Return value
  - *(integer)*: the larger of `a` and `b`.

`maxNaN( a, b )`
Returns the greater of two floating point values. If the arguments have the same value, the result is that same value. If either value is blank, then the result is blank.

- Parameters:
  - `a` *(floating point)*: an argument.
  - `b` *(floating point)*: another argument.
- Return value
  - *(floating point)*: the larger of `a` and `b`.

`maxReal( a, b )`
Returns the greater of two floating point values, ignoring blanks. If the arguments have the same value, the result is that same value. If one argument is blank, the result is the other one. If both arguments are blank, the result is blank.

Multiple-argument maximum functions are also provided in the `Arrays` and `Lists` packages.

- Parameters:
  - `a` *(floating point)*: an argument
  - `b` *(floating point)*: another argument
- Return value
  - *(floating point)*: the larger non-blank value of `a` and `b`

`min( a, b )`
Returns the smaller of two integer values. If the arguments have the same value, the result is that same value.

Multiple-argument minimum functions are also provided in the `Arrays` and `Lists` packages.

- Parameters:
  - `a` *(integer)*: an argument.
  - `b` *(integer)*: another argument.
- Return value
  - *(integer)*: the smaller of `a` and `b`.

`minNaN( a, b )`

Returns the smaller of two floating point values. If the arguments have the same value, the result is that same value. If either value is blank, then the result is blank.

- Parameters:
  - `a` *(floating point)*: an argument.
  - `b` *(floating point)*: another argument.
- Return value
  - *(floating point)*: the smaller of `a` and `b`.

`minReal( a, b )`

Returns the smaller of two floating point values, ignoring blanks. If the arguments have the same value, the result is that same value. If one argument is blank, the result is the other one. If both arguments are blank, the result is blank.

Multiple-argument minimum functions are also provided in the `Arrays` and `Lists` packages.

- Parameters:
  - `a` *(floating point)*: an argument
  - `b` *(floating point)*: another argument
- Return value
  - *(floating point)*: the larger non-blank value of `a` and `b`

`mod( a, b )`

Returns the non-negative remainder of `a/b`. This is a modulo operation, but differs from the expression `a%b` in that the answer is always >=0 (as long as `b` is not zero).

- Parameters:
  - `a` *(floating point)*: dividend
  - `b` *(floating point)*: divisor
- Return value
  - *(floating point)*: non-negative remainder when dividing `a` by `b`
- Examples:
  - `modulo(14, 5) = 4`
  - `modulo(-14, 5) = 1`
  - `modulo(2.75, 0.5) = 0.25`

`phase( t, period )`

Returns the phase of a value within a period.

For positive period, the returned value is in the range [0,1).

- Parameters:
  - `t` *(floating point)*: value
  - `period` *(floating point)*: folding period
- Return value
  - *(floating point)*: mod(t,period)/period
- Examples:
  - `phase(7, 4) = 0.75`
  - `phase(-1000.5, 2.5) = 0.8`
  - `phase(-3300, 33) = 0`

`phase( t, period, t0 )`

Returns the phase of an offset value within a period. The reference value `t0` corresponds to phase zero.

For positive period, the returned value is in the range [0,1).

- Parameters:
  - `t` *(floating point)*: value
  - `period` *(floating point)*: folding period
  - `t0` *(floating point)*: reference value, corresponding to phase zero
- Return value
  - *(floating point)*: phase(t-t0, period)
- Examples:
  - `phase(5003,100,0) = 0.03`
  - `phase(5003,100,2) = 0.01`
  - `phase(5003,100,4) = 0.99`

`phase( t, period, t0, phase0 )`

Returns the offset phase of an offset value within a period. The reference value `t0` corresponds to integer phase value, and the phase offset `phase0` determines the starting value for the phase range.

For positive period, the returned value is in the range [`phase0`,`phase0+1`).

- Parameters:
  - `t` *(floating point)*: value
  - `period` *(floating point)*: folding period
  - `t0` *(floating point)*: reference value, corresponding to phase zero
  - `phase0` *(floating point)*: offset for phase
- Return value
  - *(floating point)*: offset phase
- Examples:
  - `phase(23,10,1,99) = 99.2`
  - `phase(8.6125,0.2,0.0125,-0.3) = 0`
  - `phase(8.6125,0.2,0.1125,-0.7) = -0.5`

10.6.2 Arrays

Functions which operate on array-valued cells. The array parameters of these functions can only be used on values which are already arrays (usually, numeric arrays). In most cases that means on values in table columns which are declared as array-valued. FITS and VOTable tables can have columns which contain array values, but other formats such as CSV cannot.

If you want to calculate aggregating functions like sum, min, max etc on multiple values which are not part of an array, it's easier to use the functions from the `Lists` class.

Note that none of these functions will calculate statistical functions over a whole column of a table.

The functions fall into a number of categories:

- Aggregating operations, which map an array value to a scalar, including `size`, `count`, `countTrue`, `maximum`, `minimum`, `sum`, `mean`, `median`, `quantile`, `stdev`, `variance`, `join`.
- Operations on one or more arrays which produce array results, including `add`, `subtract`, `multiply`, `divide`, `reciprocal`, `condition`, `slice`, `pick`. Mostly these work on any numeric array type and return floating point (double precision) values, but some of them (`slice`, `pick`) have variants for different array types.
- The function `array`, which lets you assemble a floating point array value from a list of scalar numbers. There are variants (`intArray`, `stringArray`) for some different array types.

`sum( array )`
Returns the sum of all the non-blank elements in the array. If `array` is not a numeric array, `null` is returned.

- Parameters:
  - `array` *(Object)*: array of numbers
- Return value
  - *(floating point)*: sum of all the numeric values in `array`

`mean( array )`
Returns the mean of all the non-blank elements in the array. If `array` is not a numeric array, `null` is returned.

- Parameters:
  - `array` *(Object)*: array of numbers
- Return value
  - *(floating point)*: mean of all the numeric values in `array`

`variance( array )`
Returns the population variance of all the non-blank elements in the array. If `array` is not a numeric array, `null` is returned.

- Parameters:
  - `array` *(Object)*: array of numbers
- Return value
  - *(floating point)*: variance of the numeric values in `array`

`stdev( array )`
Returns the population standard deviation of all the non-blank elements in the array. If `array` is not a numeric array, `null` is returned.

- Parameters:
  - `array` *(Object)*: array of numbers
- Return value
  - *(floating point)*: standard deviation of the numeric values in `array`

`minimum( array )`
Returns the smallest of the non-blank elements in the array. If `array` is not a numeric array, `null` is returned.

- Parameters:
  - `array` *(Object)*: array of numbers
- Return value
  - *(floating point)*: minimum of the numeric values in `array`

`maximum( array )`
Returns the largest of the non-blank elements in the array. If `array` is not a numeric array, `null` is returned.

- Parameters:
  - `array` *(Object)*: array of numbers
- Return value
  - *(floating point)*: maximum of the numeric values in `array`

`median( array )`
Returns the median of the non-blank elements in the array. If `array` is not a numeric array, `null` is returned.

- Parameters:
  - `array` *(Object)*: array of numbers
- Return value
  - *(floating point)*: median of the numeric values in `array`

`quantile( array, quant )`
Returns a quantile value of the non-blank elements in the array. Which quantile is determined by the `quant` value; values of 0, 0.5 and 1 give the minimum, median and maximum respectively. A value of 0.99 would give the 99th percentile.

- Parameters:
  - `array` *(Object)*: array of numbers
  - `quant` *(floating point)*: number in the range 0-1 deterining which quantile to calculate
- Return value

- *(floating point)*: quantile corresponding to `quant`

`size( array )`
Returns the number of elements in the array. If `array` is not an array, zero is returned.

- Parameters:
  - `array` *(Object)*: array
- Return value
  - *(integer)*: size of `array`

`count( array )`
Returns the number of non-blank elements in the array. If `array` is not an array, zero is returned.

- Parameters:
  - `array` *(Object)*: array (may or may not be numeric)
- Return value
  - *(integer)*: number of non-blank elements in `array`

`countTrue( array )`
Returns the number of true elements in an array of boolean values.

- Parameters:
  - `array` *(array of boolean)*: array of true/false values
- Return value
  - *(integer)*: number of true values in `array`

`join( array, joiner )`
Returns a string composed of concatenating all the elements of an array, separated by a joiner string. If `array` is not an array, null is returned.

- Parameters:
  - `array` *(Object)*: array of numbers or strings
  - `joiner` *(String)*: text string to interpose between adjacent elements
- Return value
  - *(String)*: string composed of `array` elements separated by `joiner` strings
- Example:
  - `join(array(1.5,2.1,-3.9), "; ") = "1.5; 2.1; -3.9"`

`dotProduct( array1, array2 )`
Returns the dot (scalar) product of two numeric arrays. If either argument is not an array, or if the arrays are not of the same length, a blank value is returned.

- Parameters:
  - `array1` *(Object)*: first array
  - `array2` *(Object)*: second array

- Return value
  - *(floating point)*: sum of element-wise products of input arrays
- Example:
  - `dotProduct(array(3,4,5), array(1,2,3)) = 26`

`add( arrayOrScalar1, arrayOrScalar2 )`

Returns the element-by-element result of adding either two numeric arrays of the same length, or an array and a scalar considered as if an array of the right length.

If the arguments are not as expected (e.g. arrays of different lengths, both scalars, not numeric) then null is returned.

- Parameters:
  - `arrayOrScalar1` *(Object)*: first numeric array/scalar
  - `arrayOrScalar2` *(Object)*: second numeric array/scalar
- Return value
  - *(array of floating point)*: element-by-element result of `arrayOrScalar1 + arrayOrScalar2`, the same length as the input array(s)
- Examples:
  - `add(array(1,2,3), array(0.1,0.2,0.3)) = [1.1, 2.2, 3.3]`
  - `add(array(1,2,3), 10) = [11,12,13]`

`subtract( arrayOrScalar1, arrayOrScalar2 )`

Returns the element-by-element result of subtracting either two numeric arrays of the same length, or an array and a scalar considered as if an array of the right length.

If the arguments are not as expected (e.g. arrays of different lengths, both scalars, not numeric) then null is returned.

- Parameters:
  - `arrayOrScalar1` *(Object)*: first numeric array/scalar
  - `arrayOrScalar2` *(Object)*: second numeric array/scalar
- Return value
  - *(array of floating point)*: element-by-element result of `arrayOrScalar1 - arrayOrScalar2`, the same length as the input array(s)
- Examples:
  - `subtract(array(1,2,3), array(0.1,0.2,0.3)) = [0.9, 1.8, 2.7]`
  - `subtract(array(1,2,3), 1.0) = [0, 1, 2]`

`multiply( arrayOrScalar1, arrayOrScalar2 )`

Returns the element-by-element result of multiplying either two numeric arrays of the same length, or an array and a scalar considered as if an array of the right length.

If the arguments are not as expected (e.g. arrays of different lengths, both scalars, not numeric) then null is returned.

- Parameters:
  - `arrayOrScalar1` *(Object)*: first numeric array/scalar
  - `arrayOrScalar2` *(Object)*: second numeric array/scalar

- Return value
  - *(array of floating point)*: element-by-element result of `arrayOrScalar1 * arrayOrScalar2`, the same length as the input array(s)
- Examples:
  - `multiply(array(1,2,3), array(2,4,6)) = [2, 8, 18]`
  - `multiply(2, array(1,2,3)) = [2, 4, 6]`

`divide( arrayOrScalar1, arrayOrScalar2 )`

Returns the element-by-element result of dividing either two numeric arrays of the same length, or an array and a scalar considered as if an array of the right length.

If the arguments are not as expected (e.g. arrays of different lengths, both scalars, not numeric) then null is returned.

- Parameters:
  - `arrayOrScalar1` *(Object)*: first numeric array/scalar
  - `arrayOrScalar2` *(Object)*: second numeric array/scalar
- Return value
  - *(array of floating point)*: element-by-element result of `arrayOrScalar1 / arrayOrScalar2`, the same length as the input array(s)
- Examples:
  - `divide(array(0,9,4), array(1,3,8)) = [0, 3, 0.5]`
  - `divide(array(50,60,70), 10) = [5, 6, 7]`

`reciprocal( array )`

Returns the result of taking the reciprocal of every element of a numeric array. If the supplied `array` argument is not a numeric array, `null` is returned.

- Parameters:
  - `array` *(Object)*: array input
- Return value
  - *(array of floating point)*: array output, the same length as the `array` parameter
- Example:
  - `reciprocal(array(1,2,0.25) = [1, 0.5, 4]`

`condition( flagArray, trueValue, falseValue )`

Maps a boolean array to a numeric array by using supplied numeric values to represent true and false values from the input array.

This has the same effect as applying the expression `outArray[i] = flagArray[i] ? trueValue : falseValue`.

- Parameters:
  - `flagArray` *(array of boolean)*: array of boolean values
  - `trueValue` *(floating point)*: output value corresponding to an input true value
  - `falseValue` *(floating point)*: output value corresponding to an input false value
- Return value
  - *(array of floating point)*: output numeric array, same length as `flagArray`

- Example:
  - `condition([true, false, true], 1, 0) = [1, 0, 1]`

`constant( n, value )`

Returns a fixed-size array filled with a given constant value.

Note: This documents the double-precision version of the routine. Corresponding routines exist for other data types (`float`, `long`, `int`, `short`, `byte`).

- Parameters:
  - `n` *(integer)*: size of output array
  - `value` *(floating point)*: value of every element in the output array
- Return value
  - *(array of floating point)*: `n`-element array with every element set to `value`
- Example:
  - `constant(5, 23.5) = [23.5, 23.5, 23.5, 23.5, 23.5]`

`slice( array, i0, i1 )`

Returns a sub-sequence of values from a given array.

The semantics are like python array slicing, though both limits have to be specified: the output array contains the sequence of elements in the input array from `i0` (inclusive) to `i1` (exclusive). If a negative value is given in either case, it is added to the length of the input array, so that -1 indicates the last element of the input array. The indices are capped at 0 and the input array length respectively, so a large positive value may be used to indicate the end of the array. If the end index is less than or equal to the start index, a zero-length array is returned.

Note: This documents the double-precision version of the routine. Corresponding routines exist for other data types (`float`, `long`, `int`, `short`, `byte`, `String`, `Object`).

- Parameters:
  - `array` *(array of floating point)*: input array
  - `i0` *(integer)*: index of first element, inclusive (may be negative to count back from the end)
  - `i1` *(integer)*: index of the last element, exclusive (may be negative to count back from the end)
- Return value
  - *(array of floating point)*: array giving the sequence of elements specified by `i0` and `i1`
- Examples:
  - `slice(array(10,11,12,13), 0, 3) = [10, 11, 12]`
  - `slice(array(10,11,12,13), -2, 999) = [12, 13]`

`pick( array, indices, ... )`

Returns a selection of elements from a given array.

The output array consists of one element selected from the input array for each of the supplied index values. If a negative value is supplied for an index value, it is added to the input array length, so that -1 indicates the last element of the input array. If the input array is null, null is returned. If any of the index values is out of the range of the extent of the input array, an error results.

Note: This documents the double-precision version of the routine. Corresponding routines exist for other data types (`float`, `long`, `int`, `short`, `byte`, `String`, `Object`).

- Parameters:
  - `array` *(array of floating point)*: input array
  - `indices` *(integer, one or more)*: one or more index into the input array (may be negative to count back from the end)
- Return value
  - *(array of floating point)*: array giving the elements specified by `indices`
- Examples:
  - `pick(array(10,11,12,13), 0, 3) = [10, 13]`
  - `pick(array(10,11,12,13), -1, -2, -3) = [13, 12, 11]`

`arrayFunc( expr, inArray )`

Returns a floating-point array resulting from applying a given function expression element-by-element to an input array. The output array is the same length as the input array.

The supplied expression can use the variable "`x`" to refer to the corresponding element of the input array, and "`i`" to refer to its (zero-based) index. The various functions and operators from the expression language can all be used, but it is currently **not** possible to reference other table column values.

If there is an error in the expression, a blank value (not an array) will be returned.

- Parameters:
  - `expr` *(String)*: expression mapping input to output array values
  - `inArray` *(Object)*: input array
- Return value
  - *(array of floating point)*: floating point array with the same number of elements as `inArray`, or null for a bad `expr`
- Examples:
  - `arrayFunc("3*x",array(0,1,2,3,NaN)) = [0, 3, 6, 9, NaN]`
  - `arrayFunc("pow(2,i)+x", array(0.5,0.5,0.5,0.5)) = [1.5, 2.5, 4.5, 8.5]`

`intArrayFunc( expr, inArray )`

Returns an integer array resulting from applying a given function expression element-by-element to an input array. The output array is the same length as the input array.

The supplied expression can use the variable "`x`" to refer to the corresponding element of the input array, and "`i`" to refer to its (zero-based) index. The various functions and operators from the expression language can all be used, but it is currently **not** possible to reference other table column values.

If there is an error in the expression, a blank value (not an array) will be returned.

- Parameters:
  - `expr` *(String)*: expression mapping input to output array values
  - `inArray` *(Object)*: input array
- Return value
  - *(array of integer)*: floating point array with the same number of elements as `inArray`, or null for a bad `expr`

- Example:
  - `intArrayFunc("-x",sequence(5)) = [0, -1, -2, -3, -4]`

`indexOf( array, item )`

Returns the position in a supplied array at which a given item appears. The result is zero-based, so if the supplied `item` is the first entry in the `array`, the return value will be zero.

If the item does not appear in the array, -1 is returned. If it appears multiple times, the index of its first appearance is returned.

If `indexOf(array, item)==n`, then `array[n]` is equal to `item`.

Note: This documents the `Object` version of the routine. Corresponding routines exist for other data types (`double`, `float`, `long`, `int`, `short`).

- Parameters:
  - `array` *(array of Object)*: array which may contain the supplied item
  - `item` *(Object)*: entry to look for in the array
- Return value
  - *(integer)*: the index of `item` in `array`, or -1
- Examples:
  - `indexOf(stringArray("QSO", "BCG", "SNR"), "BCG") = 1`
  - `indexOf(stringArray("QSO", "BCG", "SNR"), "TLA") = -1`

`sequence( n )`

Returns an integer array of a given length with the values 0, 1, 2, ...

- Parameters:
  - `n` *(integer)*: length of array
- Return value
  - *(array of integer)*: `n`-element array, (0, 1, 2, ... `n`-1)
- Example:
  - `sequence(4) = (0, 1, 2, 3)`

`array( values, ... )`

Returns a floating point numeric array built from the given arguments.

- Parameters:
  - `values` *(floating point, one or more)*: one or more array elements
- Return value
  - *(array of floating point)*: array

`intArray( values, ... )`

Returns an integer numeric array built from the given arguments.

- Parameters:
  - `values` *(integer, one or more)*: one or more array elements
- Return value

- *(array of integer)*: array

`stringArray( values, ... )`
Returns a String array built from the given arguments.

- Parameters:
  - `values` *(String, one or more)*: one or more array elements
- Return value
  - *(array of String)*: array

10.6.3 Bits

Bit manipulation functions.

Note that for bitwise AND, OR, XOR of integer values etc you can use the java bitwise operators "`&`", "`|`", "`^`".

`hasBit( value, bitIndex )`
Determines whether a given integer has a certain bit set to 1.

- Parameters:
  - `value` *(long integer)*: integer whose bits are to be tested
  - `bitIndex` *(integer)*: index of bit to be tested in range 0..63, where 0 is the least significant bit
- Return value
  - *(boolean)*: true if bit is set; more or less equivalent to `(value & 1L<<bitIndex) != 0`
- Examples:
  - `hasBit(64, 6) = true`
  - `hasBit(63, 6) = false`

`bitCount( i )`
Returns the number of set bits in the 64-bit two's complement representation of the integer argument.

- Parameters:
  - `i` *(long integer)*: integer value
- Return value
  - *(integer)*: number of "1" bits in the binary representation of `i`
- Examples:
  - `bitCount(64) = 1`
  - `bitCount(3) = 2`

`toBinary( value )`
Converts the integer argument to a binary string consisting only of 1s and 0s.

- Parameters:
  - `value` *(long integer)*: integer value
- Return value
  - *(String)*: binary representation of `value`
- Examples:
  - `toBinary(42) = "101010"`
  - `toBinary(255^7) = "11111000"`

`fromBinary( binVal )`
Converts a string representing a binary number to its integer value.

- Parameters:
  - `binVal` *(String)*: binary representation of value
- Return value
  - *(integer)*: integer value represented by binary string `binVal`
- Example:
  - `fromBinary("101010") = 42`

10.6.4 Conversions

Functions for converting between strings and numeric values.

`toString( fpVal )`
Turns a numeric value into a string.

- Parameters:
  - `fpVal` *(floating point)*: floating point numeric value
- Return value
  - *(String)*: a string representation of `fpVal`

`toString( intVal )`
Turns an integer numeric value into a string.

- Parameters:
  - `intVal` *(long integer)*: integer numeric value
- Return value
  - *(String)*: a string representation of `intVal`

`toString( charVal )`
Turns a single character value into a string.

- Parameters:
  - `charVal` *(char)*: character numeric value
- Return value

- *(String)*: a string representation of `charVal`

`toString( byteVal )`
Turns a byte value into a string.

- Parameters:
  - `byteVal` *(byte)*: byte numeric value
- Return value
  - *(String)*: a string representation of `byteVal`

`toString( booleanVal )`
Turns a boolean value into a string.

- Parameters:
  - `booleanVal` *(boolean)*: boolean value (true or false)
- Return value
  - *(String)*: a string representation of `booleanVal` ("`true`" or "`false`")

`toString( objVal )`
Turns any object value into a string. As applied to existing string values this isn't really useful, but it means that you can apply `toString` to any object value without knowing its type and get a useful return from it.

- Parameters:
  - `objVal` *(Object)*: non-primitive value
- Return value
  - *(String)*: a string representation of `objVal`

`parseByte( str )`
Attempts to interpret a string as a byte (8-bit signed integer) value. If the input string can't be interpreted in this way, a blank value will result.

- Parameters:
  - `str` *(String)*: string containing numeric representation
- Return value
  - *(byte)*: byte value of `str`

`parseShort( str )`
Attempts to interpret a string as a short (16-bit signed integer) value. If the input string can't be interpreted in this way, a blank value will result.

- Parameters:
  - `str` *(String)*: string containing numeric representation
- Return value
  - *(short integer)*: byte value of `str`

`parseInt( str )`

Attempts to interpret a string as an int (32-bit signed integer) value. If the input string can't be interpreted in this way, a blank value will result.

- Parameters:
  - `str` *(String)*: string containing numeric representation
- Return value
  - *(integer)*: byte value of `str`

`parseLong( str )`

Attempts to interpret a string as a long (64-bit signed integer) value. If the input string can't be interpreted in this way, a blank value will result.

- Parameters:
  - `str` *(String)*: string containing numeric representation
- Return value
  - *(long integer)*: byte value of `str`

`parseFloat( str )`

Attempts to interpret a string as a float (32-bit floating point) value. If the input string can't be interpreted in this way, a blank value will result.

- Parameters:
  - `str` *(String)*: string containing numeric representation
- Return value
  - *(floating point)*: byte value of `str`

`parseDouble( str )`

Attempts to interpret a string as a double (64-bit signed integer) value. If the input string can't be interpreted in this way, a blank value will result.

- Parameters:
  - `str` *(String)*: string containing numeric representation
- Return value
  - *(floating point)*: byte value of `str`

`parseBigInteger( str )`

Attempts to interpret a string as a "BigInteger" value. This can be used for working with string representations of integers that can't be stored as an unsigned 64-bit value.

The result is a `BigInteger` object, which can't be used in normal numeric expressions, but has a number of methods defined on it for comparison, arithmetic, bit manipulation etc. See the java.math.BigInteger (https://docs.oracle.com/javase/8/docs/api/java/math/BigInteger.html) javadocs for details.

- Parameters:
  - `str` *(String)*: string containing numeric representation
- Return value
  - *(BigInteger)*: BigInteger value of `str`

- Examples:
  - `parseBigInteger("-20000000000000000023").doubleValue() = -2e19`
  - `parseBigInteger("18446744073709551616").testBit(64) = true`

`parseBigDecimal( str )`

Attempts to interpret a string as a "BigDecimal" value. This can be used for working with string representations of non-integer values that require more precision or range than is possible in a 64-bit IEEE-754 double precision variable.

The result is a `BigDecimal` object, which can't be used in normal numeric expressions, but has a number of methods defined on it for comparison, arithmetic, bit manipulation etc. See the java.math.BigDecimal (https://docs.oracle.com/javase/8/docs/api/java/math/BigDecimal.html) javadocs for details.

- Parameters:
  - `str` *(String)*: string contining numeric representation
- Return value
  - *(BigDecimal)*: BigDecimal value of `str`
- Example:
  - `parseBigDecimal("101").compareTo(parseBigDecimal("102")) = -1`

`parseInts( str )`

Attempts to interpret a string as an array of integer values. An ad-hoc algorithm is used that tries to extract a list of integers from a string; a comma- or space-separated list of integer values will work, and other formats may or may not.

The details of this function's behaviour may change in future releases.

- Parameters:
  - `str` *(String)*: string containing a list of integer values
- Return value
  - *(array of integer)*: array of integer values
- Examples:
  - `parseInts("9 8 -23") = [9, 8, -23]`
  - `parseInts("tiddly-pom") = []`

`parseDoubles( str )`

Attempts to interpret a string as an array of floating point values. An ad-hoc algorithm is used that tries to extract a list of numeric values from a string; a comma- or space-separated list of floating point values will work, and other formats may or may not.

This function can be used as a hacky way to extract the numeric values from an STC-S (for instance ObsCore/EPNcore `s_region`) string.

The details of this function's behaviour may change in future releases.

- Parameters:
  - `str` *(String)*: string containing a list of floating point values
- Return value
  - *(array of floating point)*: array of floating point values

- Examples:
  - `parseDoubles("1.3, 99e1, NaN, -23") = [1.3, 990.0, NaN, -23.0]`
  - `parseDoubles("Polygon ICRS 0.8 2.1 9.0 2.1 6.2 8.6") = [0.8, 2.1, 9.0, 2.1, 6.2, 8.6]`
  - `parseDoubles("La la la") = []`

`toByte( value )`
Attempts to convert the numeric argument to a byte (8-bit signed integer) result. If it is out of range, a blank value will result.

- Parameters:
  - `value` *(floating point)*: numeric value for conversion
- Return value
  - *(byte)*: `value` converted to type byte

`toShort( value )`
Attempts to convert the numeric argument to a short (16-bit signed integer) result. If it is out of range, a blank value will result.

- Parameters:
  - `value` *(floating point)*: numeric value for conversion
- Return value
  - *(short integer)*: `value` converted to type short

`toInteger( value )`
Attempts to convert the numeric argument to an int (32-bit signed integer) result. If it is out of range, a blank value will result.

- Parameters:
  - `value` *(floating point)*: numeric value for conversion
- Return value
  - *(integer)*: `value` converted to type int

`toLong( value )`
Attempts to convert the numeric argument to a long (64-bit signed integer) result. If it is out of range, a blank value will result.

- Parameters:
  - `value` *(floating point)*: numeric value for conversion
- Return value
  - *(long integer)*: `value` converted to type long

`toFloat( value )`
Attempts to convert the numeric argument to a float (32-bit floating point) result. If it is out of range, a blank value will result.

- Parameters:

- `value` *(floating point)*: numeric value for conversion

- Return value
  - *(floating point)*: `value` converted to type float

`toDouble( value )`
Converts the numeric argument to a double (64-bit signed integer) result.

- Parameters:
  - `value` *(floating point)*: numeric value for conversion
- Return value
  - *(floating point)*: `value` converted to type double

`toHex( value )`
Converts the integer argument to hexadecimal form.

- Parameters:
  - `value` *(long integer)*: integer value
- Return value
  - *(String)*: hexadecimal representation of `value`
- Example:
  - `toHex(42) = "2a"`

`fromHex( hexVal )`
Converts a string representing a hexadecimal number to its integer value.

- Parameters:
  - `hexVal` *(String)*: hexadecimal representation of value
- Return value
  - *(integer)*: integer value represented by `hexVal`
- Example:
  - `fromHex("2a") = 42`

10.6.5 CoordsDegrees

Functions for angle transformations and manipulations, with angles generally in degrees. In particular, methods for translating between degrees and HH:MM:SS.S or DDD:MM:SS.S type sexagesimal representations are provided.

`degreesToDms( deg )`
Converts an angle in degrees to a formatted degrees:minutes:seconds string. No fractional part of the seconds field is given.

- Parameters:
  - `deg` *(floating point)*: angle in degrees

- Return value
  - *(String)*: DMS-format string representing `deg`

`degreesToDms( deg, secFig )`
Converts an angle in degrees to a formatted degrees:minutes:seconds string with a given number of decimal places in the seconds field.

- Parameters:
  - `deg` *(floating point)*: angle in degrees
  - `secFig` *(integer)*: number of decimal places in the seconds field
- Return value
  - *(String)*: DMS-format string representing `deg`

`degreesToHms( deg )`
Converts an angle in degrees to a formatted hours:minutes:seconds string. No fractional part of the seconds field is given.

- Parameters:
  - `deg` *(floating point)*: angle in degrees
- Return value
  - *(String)*: HMS-format string representing `deg`

`degreesToHms( deg, secFig )`
Converts an angle in degrees to a formatted hours:minutes:seconds string with a given number of decimal places in the seconds field.

- Parameters:
  - `deg` *(floating point)*: angle in degrees
  - `secFig` *(integer)*: number of decimal places in the seconds field
- Return value
  - *(String)*: HMS-format string representing `deg`

`dmsToDegrees( dms )`
Converts a formatted degrees:minutes:seconds string to an angle in degrees. Delimiters may be colon, space, characters `dm[s]`, or some others. Additional spaces and leading +/- are permitted. The :seconds part is optional.

- Parameters:
  - `dms` *(String)*: formatted DMS string
- Return value
  - *(floating point)*: angle in degrees specified by `dms`

`hmsToDegrees( hms )`
Converts a formatted hours:minutes:seconds string to an angle in degrees. Delimiters may be colon, space, characters `hm[s]`, or some others. Additional spaces and leading +/- are permitted. The :seconds part is optional.

- Parameters:

- `hms` *(String)*: formatted HMS string

- Return value
  - *(floating point)*: angle in degrees specified by `hms`

`dmsToDegrees( deg, min, sec )`

Converts degrees, minutes, seconds to an angle in degrees.

In conversions of this type, one has to be careful to get the sign right in converting angles which are between 0 and -1 degrees. This routine uses the sign bit of the `deg` argument, taking care to distinguish between +0 and -0 (their internal representations are different for floating point values). It is illegal for the `min` or `sec` arguments to be negative.

- Parameters:
  - `deg` *(floating point)*: degrees part of angle
  - `min` *(floating point)*: minutes part of angle
  - `sec` *(floating point)*: seconds part of angle
- Return value
  - *(floating point)*: specified angle in degrees

`hmsToDegrees( hour, min, sec )`

Converts hours, minutes, seconds to an angle in degrees.

In conversions of this type, one has to be careful to get the sign right in converting angles which are between 0 and -1 hours. This routine uses the sign bit of the `hour` argument, taking care to distinguish between +0 and -0 (their internal representations are different for floating point values).

- Parameters:
  - `hour` *(floating point)*: degrees part of angle
  - `min` *(floating point)*: minutes part of angle
  - `sec` *(floating point)*: seconds part of angle
- Return value
  - *(floating point)*: specified angle in degrees

`skyDistanceDegrees( ra1, dec1, ra2, dec2 )`

Calculates the separation (distance around a great circle) of two points on the sky in degrees.

- Parameters:
  - `ra1` *(floating point)*: right ascension of point 1 in degrees
  - `dec1` *(floating point)*: declination of point 1 in degrees
  - `ra2` *(floating point)*: right ascension of point 2 in degrees
  - `dec2` *(floating point)*: declination of point 2 in degrees
- Return value
  - *(floating point)*: angular distance between point 1 and point 2 in degrees

`posAngDegrees( ra1, dec1, ra2, dec2 )`

Calculates the position angle between two points on the sky in degrees. The result is in the range +/-180. If point 2 is due east of point 1, the result is +90. Zero is returned if the points are coincident.

- Parameters:
  - `ra1` *(floating point)*: right ascension of point 1 in degrees
  - `dec1` *(floating point)*: declination of point 1 in degrees
  - `ra2` *(floating point)*: right ascension of point 2 in degrees
  - `dec2` *(floating point)*: declination of point 2 in degrees
- Return value
  - *(floating point)*: bearing in degrees of point 2 from point 1.

`polarDistanceDegrees( ra1, dec1, radius1, ra2, dec2, radius2 )`

Calculates the distance in three dimensional space between two points specified in spherical polar coordinates.

- Parameters:
  - `ra1` *(floating point)*: right ascension of point 1 in degrees
  - `dec1` *(floating point)*: declination of point1 in degrees
  - `radius1` *(floating point)*: distance from origin of point1
  - `ra2` *(floating point)*: right ascension of point 2 in degrees
  - `dec2` *(floating point)*: declination of point2 in degrees
  - `radius2` *(floating point)*: distance from origin of point2
- Return value
  - *(floating point)*: the linear distance between point1 and point2; units are as for `radius1` and `radius2`

10.6.6 CoordsRadians

Functions for angle transformations and manipulations, based on radians rather than degrees. In particular, methods for translating between radians and HH:MM:SS.S or DDD:MM:SS.S type sexagesimal representations are provided.

`radiansToDms( rad )`

Converts an angle in radians to a formatted degrees:minutes:seconds string. No fractional part of the seconds field is given.

- Parameters:
  - `rad` *(floating point)*: angle in radians
- Return value
  - *(String)*: DMS-format string representing `rad`

`radiansToDms( rad, secFig )`

Converts an angle in radians to a formatted degrees:minutes:seconds string with a given number of decimal places in the seconds field.

- Parameters:
  - `rad` *(floating point)*: angle in radians
  - `secFig` *(integer)*: number of decimal places in the seconds field
- Return value
  - *(String)*: DMS-format string representing `rad`

`radiansToHms( rad )`
Converts an angle in radians to a formatted hours:minutes:seconds string. No fractional part of the seconds field is given.

- Parameters:
  - `rad` *(floating point)*: angle in radians
- Return value
  - *(String)*: HMS-format string representing `rad`

`radiansToHms( rad, secFig )`
Converts an angle in radians to a formatted hours:minutes:seconds string with a given number of decimal places in the seconds field.

- Parameters:
  - `rad` *(floating point)*: angle in radians
  - `secFig` *(integer)*: number of decimal places in the seconds field
- Return value
  - *(String)*: HMS-format string representing `rad`

`dmsToRadians( dms )`
Converts a formatted degrees:minutes:seconds string to an angle in radians. Delimiters may be colon, space, characters `dm[s]`, or some others. Additional spaces and leading +/- are permitted. The :seconds part is optional.

- Parameters:
  - `dms` *(String)*: formatted DMS string
- Return value
  - *(floating point)*: angle in radians specified by `dms`

`hmsToRadians( hms )`
Converts a formatted hours:minutes:seconds string to an angle in radians. Delimiters may be colon, space, characters `hm[s]`, or some others. Additional spaces and leading +/- are permitted. The :seconds part is optional.

- Parameters:
  - `hms` *(String)*: formatted HMS string
- Return value
  - *(floating point)*: angle in radians specified by `hms`

`dmsToRadians( deg, min, sec )`
Converts degrees, minutes, seconds to an angle in radians.

In conversions of this type, one has to be careful to get the sign right in converting angles which are between 0 and -1 degrees. This routine uses the sign bit of the `deg` argument, taking care to distinguish between +0 and -0 (their internal representations are different for floating point values). It is illegal for the `min` or `sec` arguments to be negative.

- Parameters:

- `deg` *(floating point)*: degrees part of angle
- `min` *(floating point)*: minutes part of angle
- `sec` *(floating point)*: seconds part of angle

- Return value
  - *(floating point)*: specified angle in radians

`hmsToRadians( hour, min, sec )`
Converts hours, minutes, seconds to an angle in radians.

In conversions of this type, one has to be careful to get the sign right in converting angles which are between 0 and -1 hours. This routine uses the sign bit of the `hour` argument, taking care to distinguish between +0 and -0 (their internal representations are different for floating point values).

- Parameters:
  - `hour` *(floating point)*: degrees part of angle
  - `min` *(floating point)*: minutes part of angle
  - `sec` *(floating point)*: seconds part of angle
- Return value
  - *(floating point)*: specified angle in radians

`skyDistanceRadians( ra1, dec1, ra2, dec2 )`
Calculates the separation (distance around a great circle) of two points on the sky in radians.

- Parameters:
  - `ra1` *(floating point)*: right ascension of point 1 in radians
  - `dec1` *(floating point)*: declination of point 1 in radians
  - `ra2` *(floating point)*: right ascension of point 2 in radians
  - `dec2` *(floating point)*: declination of point 2 in radians
- Return value
  - *(floating point)*: angular distance between point 1 and point 2 in radians

`posAngRadians( ra1, dec1, ra2, dec2 )`
Calculates the position angle between two points on the sky in radians. The result is in the range +/-pi. If point 2 is due east of point 1, the result is +pi/2. Zero is returned if the points are coincident.

- Parameters:
  - `ra1` *(floating point)*: right ascension of point 1 in radians
  - `dec1` *(floating point)*: declination of point 1 in radians
  - `ra2` *(floating point)*: right ascension of point 2 in radians
  - `dec2` *(floating point)*: declination of point 2 in radians
- Return value
  - *(floating point)*: bearing in radians of point 2 from point 1

`polarDistanceRadians( ra1, dec1, radius1, ra2, dec2, radius2 )`
Calculates the distance in three dimensional space between two points specified in spherical polar coordinates.

- Parameters:

- `ra1` *(floating point)*: right ascension of point 1 in radians
- `dec1` *(floating point)*: declination of point1 in radians
- `radius1` *(floating point)*: distance from origin of point1
- `ra2` *(floating point)*: right ascension of point 2 in radians
- `dec2` *(floating point)*: declination of point2 in radians
- `radius2` *(floating point)*: distance from origin of point2

- Return value
  - *(floating point)*: the linear distance between point1 and point2; units are as for `radius1` and `radius2`

`hoursToRadians( hours )`
Converts hours to radians.

- Parameters:
  - `hours` *(floating point)*: angle in hours
- Return value
  - *(floating point)*: angle in radians

`degreesToRadians( deg )`
Converts degrees to radians.

- Parameters:
  - `deg` *(floating point)*: angle in degrees
- Return value
  - *(floating point)*: angle in radians

`radiansToDegrees( rad )`
Converts radians to degrees.

- Parameters:
  - `rad` *(floating point)*: angle in radians
- Return value
  - *(floating point)*: angle in degrees

`raFK4toFK5radians( raFK4, decFK4 )`
Converts a B1950.0 FK4 position to J2000.0 FK5 at an epoch of B1950.0 yielding Right Ascension. This assumes zero proper motion in the FK5 frame.

- Parameters:
  - `raFK4` *(floating point)*: right ascension in B1950.0 FK4 system (radians)
  - `decFK4` *(floating point)*: declination in B1950.0 FK4 system (radians)
- Return value
  - *(floating point)*: right ascension in J2000.0 FK5 system (radians)

`decFK4toFK5radians( raFK4, decFK4 )`
Converts a B1950.0 FK4 position to J2000.0 FK5 at an epoch of B1950.0 yielding Declination This assumes zero proper motion in the FK5 frame.

- Parameters:
  - `raFK4` *(floating point)*: right ascension in B1950.0 FK4 system (radians)
  - `decFK4` *(floating point)*: declination in B1950.0 FK4 system (radians)
- Return value
  - *(floating point)*: declination in J2000.0 FK5 system (radians)

`raFK5toFK4radians( raFK5, decFK5 )`

Converts a J2000.0 FK5 position to B1950.0 FK4 at an epoch of B1950.0 yielding Declination. This assumes zero proper motion, parallax and radial velocity in the FK5 frame.

- Parameters:
  - `raFK5` *(floating point)*: right ascension in J2000.0 FK5 system (radians)
  - `decFK5` *(floating point)*: declination in J2000.0 FK5 system (radians)
- Return value
  - *(floating point)*: right ascension in the FK4 system (radians)

`decFK5toFK4radians( raFK5, decFK5 )`

Converts a J2000.0 FK5 position to B1950.0 FK4 at an epoch of B1950.0 yielding Declination. This assumes zero proper motion, parallax and radial velocity in the FK5 frame.

- Parameters:
  - `raFK5` *(floating point)*: right ascension in J2000.0 FK5 system (radians)
  - `decFK5` *(floating point)*: declination in J2000.0 FK5 system (radians)
- Return value
  - *(floating point)*: right ascension in the FK4 system (radians)

`raFK4toFK5Radians( raFK4, decFK4, bepoch )`

Converts a B1950.0 FK4 position to J2000.0 FK5 yielding Right Ascension. This assumes zero proper motion in the FK5 frame. The `bepoch` parameter is the epoch at which the position in the FK4 frame was determined.

- Parameters:
  - `raFK4` *(floating point)*: right ascension in B1950.0 FK4 system (radians)
  - `decFK4` *(floating point)*: declination in B1950.0 FK4 system (radians)
  - `bepoch` *(floating point)*: Besselian epoch
- Return value
  - *(floating point)*: right ascension in J2000.0 FK5 system (radians)

`decFK4toFK5Radians( raFK4, decFK4, bepoch )`

Converts a B1950.0 FK4 position to J2000.0 FK5 yielding Declination. This assumes zero proper motion in the FK5 frame. The `bepoch` parameter is the epoch at which the position in the FK4 frame was determined.

- Parameters:
  - `raFK4` *(floating point)*: right ascension in B1950.0 FK4 system (radians)
  - `decFK4` *(floating point)*: declination in B1950.0 FK4 system (radians)
  - `bepoch` *(floating point)*: Besselian epoch

- Return value
  - *(floating point)*: declination in J2000.0 FK5 system (radians)

raFK5toFK4Radians(raFK5, decFK5, bepoch)
Converts a J2000.0 FK5 position to B1950.0 FK4 yielding Declination. This assumes zero proper motion, parallax and radial velocity in the FK5 frame.

- Parameters:
  - `raFK5` *(floating point)*: right ascension in J2000.0 FK5 system (radians)
  - `decFK5` *(floating point)*: declination in J2000.0 FK5 system (radians)
  - `bepoch` *(floating point)*: Besselian epoch
- Return value
  - *(floating point)*: right ascension in the FK4 system (radians)

decFK5toFK4Radians(raFK5, decFK5, bepoch)
Converts a J2000.0 FK5 position to B1950.0 FK4 yielding Declination. This assumes zero proper motion, parallax and radial velocity in the FK5 frame.

- Parameters:
  - `raFK5` *(floating point)*: right ascension in J2000.0 FK5 system (radians)
  - `decFK5` *(floating point)*: declination in J2000.0 FK5 system (radians)
  - `bepoch` *(floating point)*: Besselian epoch
- Return value
  - *(floating point)*: right ascension in the FK4 system (radians)

DEGREE_RADIANS
The size of one degree in radians.

HOUR_RADIANS
The size of one hour of right ascension in radians.

ARC_MINUTE_RADIANS
The size of one arcminute in radians.

ARC_SECOND_RADIANS
The size of one arcsecond in radians.

10.6.7 Coverage

Functions related to coverage and footprints.

One coverage standard is Multi-Order Coverage maps, described at http://www.ivoa.net/Documents/MOC/ (http://www.ivoa.net/Documents/MOC/). MOC positions are always defined in ICRS equatorial coordinates.

MOC locations may be given as either the filename or the URL of a MOC FITS file. Alternatively,

they may be the identifier of a VizieR table, for instance "`V/139/sdss9`" (SDSS DR9). A list of all the MOCs available from VizieR can currently be found at http://alasky.u-strasbg.fr/footprints/tables/vizier/ (http://alasky.u-strasbg.fr/footprints/tables/vizier/). You can search for VizieR table identifiers from the VizieR web page (http://vizier.u-strasbg.fr/ (http://vizier.u-strasbg.fr/)); note you must use the *table* identifier (like "`V/139/sdss9`") and not the *catalogue* identifier (like "`V/139`").

`inMoc( mocLocation, ra, dec )`

Indicates whether a given sky position falls strictly within a given MOC (Multi-Order Coverage map). If the given `mocLocation` value does not represent a MOC (for instance no file exists or the file is not in MOC format) a warning will be issued the first time it's referenced, and the result will be false.

- Parameters:
  - `mocLocation` *(String)*: location of a FITS MOC file: a filename, a URL, or a VizieR table name
  - `ra` *(floating point)*: ICRS right ascension in degrees
  - `dec` *(floating point)*: ICRS declination in degrees
- Return value
  - *(boolean)*: true iff the given position falls within the given MOC

`nearMoc( mocLocation, ra, dec, distanceDeg )`

Indicates whether a given sky position either falls within, or is within a certain distance of the edge of, a given MOC (Multi-Order Coverage map). If the given `mocLocation` value does not represent a MOC (for instance no file exists or the file is not in MOC format) a warning will be issued the first time it's referenced, and the result will be false.

- Parameters:
  - `mocLocation` *(String)*: location of a FITS MOC file: a filename, a URL, or a VizieR table name
  - `ra` *(floating point)*: ICRS right ascension in degrees
  - `dec` *(floating point)*: ICRS declination in degrees
  - `distanceDeg` *(floating point)*: permitted distance from MOC boundary in degrees
- Return value
  - *(boolean)*: true iff the given position is within `distance` degrees of the given MOC

10.6.8 Distances

Functions for converting between different measures of cosmological distance.

The following parameters are used:

- **z**: redshift
- **H0**: Hubble constant in km/sec/Mpc (example value ~70)
- **omegaM**: Density ratio of the universe (example value 0.3)
- **omegaLambda**: Normalised cosmological constant (example value 0.7)

For a flat universe, `omegaM+omegaLambda=1`

The terms and formulae used here are taken from the paper by D.W.Hogg, *Distance measures in*

cosmology, astro-ph/9905116 (http://arxiv.org/abs/astro-ph/9905116) v4 (2000).

`MpcToM( distMpc )`
Converts from MegaParsecs to metres.

- Parameters:
  - `distMpc` *(floating point)*: distance in Mpc
- Return value
  - *(floating point)*: distance in m

`mToMpc( distM )`
Converts from metres to MegaParsecs.

- Parameters:
  - `distM` *(floating point)*: distance in m
- Return value
  - *(floating point)*: distance in Mpc

`zToDist( z )`
Quick and dirty function for converting from redshift to distance.

Warning: this makes some reasonable assumptions about the cosmology and returns the luminosity distance. It is only intended for approximate use. If you care about the details, use one of the more specific functions here.

- Parameters:
  - `z` *(floating point)*: redshift
- Return value
  - *(floating point)*: some distance measure in Mpc

`zToAge( z )`
Quick and dirty function for converting from redshift to time.

Warning: this makes some reasonable assumptions about the cosmology. It is only intended for approximate use. If you care about the details use one of the more specific functions here.

- Parameters:
  - `z` *(floating point)*: redshift
- Return value
  - *(floating point)*: 'age' of photons from redshift `z` in Gyr

`comovingDistanceL( z, H0, omegaM, omegaLambda )`
Line-of-sight comoving distance.

- Parameters:
  - `z` *(floating point)*: redshift
  - `H0` *(floating point)*: Hubble constant in km/sec/Mpc
  - `omegaM` *(floating point)*: density ratio of the universe
  - `omegaLambda` *(floating point)*: normalised cosmological constant

- Return value
  - *(floating point)*: line-of-sight comoving distance in Mpc

`comovingDistanceT( z, H0, omegaM, omegaLambda )`
Transverse comoving distance.

- Parameters:
  - `z` *(floating point)*: redshift
  - `H0` *(floating point)*: Hubble constant in km/sec/Mpc
  - `omegaM` *(floating point)*: density ratio of the universe
  - `omegaLambda` *(floating point)*: normalised cosmological constant
- Return value
  - *(floating point)*: transverse comoving distance in Mpc

`angularDiameterDistance( z, H0, omegaM, omegaLambda )`
Angular diameter distance.

- Parameters:
  - `z` *(floating point)*: redshift
  - `H0` *(floating point)*: Hubble constant in km/sec/Mpc
  - `omegaM` *(floating point)*: density ratio of the universe
  - `omegaLambda` *(floating point)*: normalised cosmological constant
- Return value
  - *(floating point)*: angular diameter distance in Mpc

`luminosityDistance( z, H0, omegaM, omegaLambda )`
Luminosity distance.

- Parameters:
  - `z` *(floating point)*: redshift
  - `H0` *(floating point)*: Hubble constant in km/sec/Mpc
  - `omegaM` *(floating point)*: density ratio of the universe
  - `omegaLambda` *(floating point)*: normalised cosmological constant
- Return value
  - *(floating point)*: luminosity distance in Mpc

`lookbackTime( z, H0, omegaM, omegaLambda )`
Lookback time. This returns the difference between the age of the universe at time of observation (now) and the age of the universe at the time when photons of redshift `z` were emitted.

- Parameters:
  - `z` *(floating point)*: redshift
  - `H0` *(floating point)*: Hubble constant in km/sec/Mpc
  - `omegaM` *(floating point)*: density ratio of the universe
  - `omegaLambda` *(floating point)*: normalised cosmological constant
- Return value
  - *(floating point)*: lookback time in Gyr

comovingVolume(z, H0, omegaM, omegaLambda)
Comoving volume. This returns the all-sky total comoving volume out to a given redshift `z`.

- Parameters:
  - `z` *(floating point)*: redshift
  - `H0` *(floating point)*: Hubble constant in km/sec/Mpc
  - `omegaM` *(floating point)*: density ratio of the universe
  - `omegaLambda` *(floating point)*: normalised cosmological constant
- Return value
  - *(floating point)*: comoving volume in Gpc^3

SPEED_OF_LIGHT
Speed of light in m/s.

METRE_PER_PARSEC
Number of metres in a parsec.

SEC_PER_YEAR
Number of seconds in a year.

10.6.9 Fluxes

Functions for conversion between flux and magnitude values. Functions are provided for conversion between flux in Janskys and AB magnitudes.

Some constants for approximate conversions between different magnitude scales are also provided:

- Constants `JOHNSON_AB_*`, for Johnson <-> AB magnitude conversions, from Frei and Gunn, Astronomical Journal *108*, 1476 (1994), Table 2 (1994AJ....108.1476F (http://adsabs.harvard.edu/abs/1994AJ....108.1476F)).
- Constants `VEGA_AB_*`, for Vega <-> AB magnitude conversions, from Blanton et al., Astronomical Journal *129*, 2562 (2005), Eqs. (5) (2005AJ....129.2562B (http://adsabs.harvard.edu/abs/2005AJ....129.2562B)).

abToJansky(magAB)
Converts AB magnitude to flux in Jansky.
$F/Jy=10^{(23-(AB+48.6)/2.5)}$

- Parameters:
  - `magAB` *(floating point)*: AB magnitude value
- Return value
  - *(floating point)*: equivalent flux in Jansky

janskyToAb(fluxJansky)
Converts flux in Jansky to AB magnitude.
$AB=2.5*(23-\log_{10}(F/Jy))-48.6$

- Parameters:
  - `fluxJansky` *(floating point)*: flux in Jansky
- Return value
  - *(floating point)*: equivalent AB magnitude

`luminosityToFlux( lumin, dist )`
Converts luminosity to flux given a luminosity distance.

F=lumin/(4 x Pi x $dist^2$)

- Parameters:
  - `lumin` *(floating point)*: luminosity
  - `dist` *(floating point)*: luminosity distance
- Return value
  - *(floating point)*: equivalent flux

`fluxToLuminosity( flux, dist )`
Converts flux to luminosity given a luminosity distance.

lumin=(4 x Pi x $dist^2$) F

- Parameters:
  - `flux` *(floating point)*: flux
  - `dist` *(floating point)*: luminosity distance
- Return value
  - *(floating point)*: equivalent luminosity

`JOHNSON_AB_V`
Approximate offset between Johnson and AB magnitudes in V band. $V_J \sim = V_{AB} +$`JOHNSON_AB_V`.

`JOHNSON_AB_B`
Approximate offset between Johnson and AB magnitudes in B band. $B_J \sim = B_{AB} +$`JOHNSON_AB_B`.

`JOHNSON_AB_Bj`
Approximate offset between Johnson and AB magnitudes in Bj band. $Bj_J \sim = Bj_{AB} +$`JOHNSON_AB_Bj`.

`JOHNSON_AB_R`
Approximate offset between Johnson and AB magnitudes in R band. $R_J \sim = R_{AB} +$`JOHNSON_AB_R`.

`JOHNSON_AB_I`
Approximate offset between Johnson and AB magnitudes in I band. $I_J \sim = I_{AB} +$`JOHNSON_AB_I`.

`JOHNSON_AB_g`

Approximate offset between Johnson and AB magnitudes in g band. g_J~=g_{AB}+JOHNSON_AB_g.

JOHNSON_AB_r
Approximate offset between Johnson and AB magnitudes in r band. r_J~=r_{AB}+JOHNSON_AB_r.

JOHNSON_AB_i
Approximate offset between Johnson and AB magnitudes in i band. i_J~=i_{AB}+JOHNSON_AB_i.

JOHNSON_AB_Rc
Approximate offset between Johnson and AB magnitudes in Rc band. Rc_J~=Rc_{AB}+JOHNSON_AB_Rc.

JOHNSON_AB_Ic
Approximate offset between Johnson and AB magnitudes in Ic band. Ic_J~=Ic_{AB}+JOHNSON_AB_Ic.

JOHNSON_AB_uPrime
Offset between Johnson and AB magnitudes in u' band (zero). u'_J=u'_{AB}+JOHNSON_AB_uPrime=u'_{AB}.

JOHNSON_AB_gPrime
Offset between Johnson and AB magnitudes in g' band (zero). g'_J=g'_{AB}+JOHNSON_AB_gPrime=g'_{AB}.

JOHNSON_AB_rPrime
Offset between Johnson and AB magnitudes in r' band (zero). r'_J=r'_{AB}+JOHNSON_AB_rPrime=r'AB.

JOHNSON_AB_iPrime
Offset between Johnson and AB magnitudes in i' band (zero). i'_J=i'_{AB}+JOHNSON_AB_iPrime=i'AB.

JOHNSON_AB_zPrime
Offset between Johnson and AB magnitudes in z' band (zero). z'_J=z'_{AB}+JOHNSON_AB_zPrime=z'_{AB}.

VEGA_AB_J
Approximate offset between Vega (as in 2MASS) and AB magnitudes in J band. J_{Vega}~=J_{AB}+VEGA_AB_J.

VEGA_AB_H
Approximate offset between Vega (as in 2MASS) and AB magnitudes in H band. H_{Vega}~=HAB+VEGA_AB_H.

VEGA_AB_K

Approximate offset between Vega (as in 2MASS) and AB magnitudes in K band. K_{Vega}~=K_{AB}+`VEGA_AB_K`.

10.6.10 Formats

Functions for formatting numeric values.

`formatDecimal( value, dp )`
Turns a floating point value into a string with a given number of decimal places using standard settings.

- Parameters:
  - `value` *(floating point)*: value to format
  - `dp` *(integer)*: number of decimal places (digits after the decmal point)
- Return value
  - *(String)*: formatted string
- Examples:
  - `formatDecimal(PI,0) = "3."`
  - `formatDecimal(0,10) = ".0000000000"`
  - `formatDecimal(E*10,3) = "27.183"`

`formatDecimalLocal( value, dp )`
Turns a floating point value into a string using current locale settings. For instance if language is set to French, decimal points will be represented as a comma "," instead of a full stop ".". Otherwise behaves the same as the corresponding `formatDecimal` function.

- Parameters:
  - `value` *(floating point)*: value to format
  - `dp` *(integer)*: number of decimal places (digits after the decmal point)
- Return value
  - *(String)*: formatted string
- Examples:
  - `formatDecimal(PI,0) = "3,"`
  - `formatDecimal(0,10) = ",0000000000"`
  - `formatDecimal(E*10,3) = "27,183"`

`formatDecimal( value, format )`
Turns a floating point value into a formatted string using standard settings. The `format` string is as defined by Java's `java.text.DecimalFormat` (http://docs.oracle.com/javase/6/docs/api/java/text/DecimalFormat.html) class.

- Parameters:
  - `value` *(floating point)*: value to format
  - `format` *(String)*: format specifier
- Return value
  - *(String)*: formatted string

- Examples:
  - `formatDecimal(99, "#.000") = "99.000"`
  - `formatDecimal(PI, "+0.##;-0.##") = "+3.14"`

`formatDecimalLocal( value, format )`

Turns a floating point value into a formatted string using current locale settings. For instance if language is set to French, decimal points will be represented as a comma "," instead of a full stop ".". Otherwise behaves the same as the corresponding `formatDecimal` function.

- Parameters:
  - `value` *(floating point)*: value to format
  - `format` *(String)*: format specifier
- Return value
  - *(String)*: formatted string
- Examples:
  - `formatDecimal(99, "#.000") = "99,000"`
  - `formatDecimal(PI, "+0.##;-0.##") = "+3,14"`

10.6.11 Gaia

Functions related to astrometry suitable for use with data from the Gaia astrometry mission.

The methods here are not specific to the Gaia mission, but the parameters of the functions and their units are specified in a form that is convenient for use with Gaia data, in particular the `gaia_source` catalogue available from http://gea.esac.esa.int/archive/ (http://gea.esac.esa.int/archive/) and copies or mirrors.

There are currently three main sets of functions here:

- position and velocity vector calculation and manipulation
- distance estimation from parallaxes
- astrometry propagation to different epochs

Position and velocity vectors

Functions are provided for converting the astrometric parameters contained in the Gaia catalogue to ICRS Cartesian position (XYZ) and velocity (UVW) vectors. Functions are also provided to convert these vectors between ICRS and Galactic or Ecliptic coordinates. The calculations are fairly straightforward, and follow the equations laid out in section 1.5.6 of *The Hipparcos and Tycho Catalogues*, ESA SP-1200 (https://www.cosmos.esa.int/web/hipparcos/catalogues) (1997) and also section 3.1.7 of the Gaia DR2 documentation (http://gea.esac.esa.int/archive/documentation/GDR2/) (2018).

These functions will often be combined; for instance to calculate the position and velocity in galactic coordinates from Gaia catalogue values, the following expressions may be useful:

```
xyz_gal = icrsToGal(astromXYZ(ra,dec,parallax))
uvw_gal = icrsToGal(astromUVW(array(ra,dec,parallax,pmra,pmdec,radial_velocity)))
```

though note that these particular examples simply invert parallax to provide distance estimates, which is not generally valid. Note also that these functions do not attempt to correct for solar

motion. Such adjustments should be carried out by hand on the results of these functions if they are required.

Functions for calculating errors on the Cartesian components based on the error and correlation quantities from the Gaia catalogue are not currently provided. They would require fairly complicated invocations. If there is demand they may be implemented in the future.

Distance estimation

Gaia measures parallaxes, but some scientific use cases require the radial distance instead. While distance in parsec is in principle the reciprocal of parallax in arcsec, in the presence of non-negligable errors on measured parallax, this inversion does not give a good estimate of distance. A thorough discussion of this topic and approaches to estimating distances for Gaia-like data can be found in the papers

- C.A.L.Bailer-Jones, "*Estimating distances from parallaxes*", PASP *127*, p994 (2015) 2015PASP..127..994B (http://adsabs.harvard.edu/abs/2015PASP..127..994B)
- T.L.Astraatmadja and C.A.L.Bailer-Jones, "*Estimating Distances from Parallaxes. II. Performance of Bayesian Distance Estimators on a Gaia-like Catalogue*", ApJ *832*, a137 (2016) 2016ApJ...832..137A (http://adsabs.harvard.edu/abs/2016ApJ...832..137A)
- X.Luri et al., "*Gaia Data Release 2: Using Gaia Parallaxes*", A&A *in press* (2018) arXiv:1804.09376 (https://arxiv.org/abs/1804.09376)

The functions provided here correspond to calculations from Astraatmadja & Bailer-Jones, "*Estimating Distances from Parallaxes. III. Distances of Two Million Stars in the Gaia DR1 Catalogue*", ApJ *833*, a119 (2016) 2016ApJ...833..119A (http://adsabs.harvard.edu/abs/2016ApJ...833..119A) based on the **Exponentially Decreasing Space Density** prior defined therein. This implementation was written with reference to the Java implementation by Enrique Utrilla (DPAC).

These functions are parameterised by a length scale *L* that defines the exponential decay (the mode of the prior PDF is at $r=2L$). Some value for this length scale, specified in parsec, must be supplied to the functions as the `lpc` parameter.

Note that the values provided by these functions do *not* match those from the paper Bailer-Jones et al. "*Estimating Distances from Parallaxes IV: Distances to 1.33 Billion stars in Gaia Data Release 2"*, accepted for AJ (2018) arXiv:1804.10121 (https://arxiv.org/abs/1804.10121). The calculations of that paper differ from the ones presented here in several ways: it uses a galactic model for the direction-dependent length scale not currently available here, it pre-applies a parallax correction of -0.029mas, and it uses different uncertainty measures and in some cases (bimodal PDF) a different best distance estimator.

Epoch Propagation

The Gaia source catalogue provides, for at least some sources, the six-parameter astrometric solution (Right Ascension, Declination, Parallax, Proper motion in RA and Dec, and Radial Velocity), along with errors on these values and correlations between these errors. While a crude estimate of the position at an earlier or later epoch than that of the measurement can be made by multiplying the proper motion components by epoch difference and adding to the measured position, a more careful treatment is required for accurate propagation between epochs of the astrometric parameters, and if required their errors and correlations. The expressions for this are set out in section 1.5.5 (Volume 1) of *The Hipparcos and Tycho Catalogues*, ESA SP-1200 (https://www.cosmos.esa.int/web/hipparcos/catalogues) (1997) (but see below), and the code is based on an implementation by Alexey Butkevich and Daniel Michalik (DPAC). A correction is applied to the SP-1200 treatment of radial velocity uncertainty following *Michalik et al. 2014* 2014A&A...571A..85M (http://ukads.nottingham.ac.uk/abs/2014A%26A...571A..85M) because of

their better handling of small radial velocities or parallaxes.

The calculations give the same results, though not exactly in the same form, as the epoch propagation functions available in the Gaia archive service.

`polarXYZ( phi, theta, r )`
Converts from spherical polar to Cartesian coordinates.

- Parameters:
  - `phi` *(floating point)*: longitude in degrees
  - `theta` *(floating point)*: latitude in degrees
  - `r` *(floating point)*: radial distance
- Return value
  - *(array of floating point)*: 3-element vector giving Cartesian coordinates
- Examples:
  - `polarXYZ(ra, dec, distance_estimate)`
  - `polarXYZ(l, b, 3262./parallax)` - calculates vector components in units of light year in the galactic system, on the assumption that distance is the inverse of parallax

`astromXYZ( ra, dec, parallax )`
Calculates Cartesian components of position from RA, Declination and parallax. This is a convenience function, equivalent to:

```
polarXYZ(ra, dec, 1000./parallax)
```

Note that this performs distance scaling using a simple inversion of parallax, which is not in general reliable for parallaxes with non-negligable errors. Use at your own risk.

- Parameters:
  - `ra` *(floating point)*: Right Ascension in degrees
  - `dec` *(floating point)*: Declination in degrees
  - `parallax` *(floating point)*: parallax in mas
- Return value
  - *(array of floating point)*: 3-element vector giving equatorial space coordinates in parsec
- Examples:
  - `astromXYZ(ra, dec, parallax)`
  - `icrsToGal(astromXYZ(ra, dec, parallax))`

`icrsToGal( xyz )`
Converts a 3-element vector representing ICRS (equatorial) coordinates to galactic coordinates. This can be used with position or velocity vectors.

The input vector is multiplied by the matrix $\mathbf{A_G}'$, given in Eq. 3.61 of the Gaia DR2 documentation, following Eq. 1.5.13 of the Hipparcos catalogue.

The output coordinate system is right-handed, with the three components positive in the directions of the Galactic center, Galactic rotation, and the North Galactic Pole respectively.

- Parameters:

- `xyz` *(array of floating point)*: 3-element vector giving ICRS Cartesian components

- Return value
  - *(array of floating point)*: 3-element vector giving Galactic Cartesian components
- Example:
  - `icrsToGal(polarXYZ(ra, dec, distance))`

`galToIcrs( xyz )`

Converts a 3-element vector representing galactic coordinates to ICRS (equatorial) coordinates. This can be used with position or velocity vectors.

The input vector is multiplied by the matrix $\mathbf{A_G}$, given in Eq. 3.61 of the Gaia DR2 documentation, following Eq. 1.5.13 of the Hipparcos catalogue.

The input coordinate system is right-handed, with the three components positive in the directions of the Galactic center, Galactic rotation, and the North Galactic Pole respectively.

- Parameters:
  - `xyz` *(array of floating point)*: 3-element vector giving Galactic Cartesian components
- Return value
  - *(array of floating point)*: 3-element vector giving ICRS Cartesian components
- Example:
  - `galToIcrs(polarXYZ(l, b, distance))`

`icrsToEcl( xyz )`

Converts a 3-element vector representing ICRS (equatorial) coordinates to ecliptic coordinates. This can be used with position or velocity vectors.

The transformation corresponds to that between the coordinates `(ra,dec)` and `(ecl_lon,ecl_lat)` in the Gaia source catalogue (DR2).

- Parameters:
  - `xyz` *(array of floating point)*: 3-element vector giving ICRS Cartesian components
- Return value
  - *(array of floating point)*: 3-element vector giving ecliptic Cartesian components
- Example:
  - `icrsToEcl(polarXYZ(ra, dec, distance))`

`eclToIcrs( xyz )`

Converts a 3-element vector representing ecliptic coordinates to ICRS (equatorial) coordinates. This can be used with position or velocity vectors.

The transformation corresponds to that between the coordinates `(ecl_lon,ecl_lat)` and `(ra,dec)` in the Gaia source catalogue (DR2).

- Parameters:
  - `xyz` *(array of floating point)*: 3-element vector giving ecliptic Cartesian coordinates
- Return value
  - *(array of floating point)*: 3-element vector giving ICRS Cartesian coordinates

- Example:
  - `eclToIcrs(polarXYZ(ecl_lon, ecl_lat, distance))`

`astromUVW( astrom6 )`

Calculates Cartesian components of velocity from quantities available in the Gaia source catalogue. The output is in the same coordinate system as the inputs, that is ICRS for the correspondingly-named Gaia quantities.

The input astrometry parameters are represented by a 6-element array, with the following elements:

index	gaia_source name	unit	description
0:	ra	deg	right ascension
1:	dec	deg	declination
2:	parallax	mas	parallax
3:	pmra	mas/yr	proper motion in ra * cos(dec)
4:	pmdec	mas/yr	proper motion in dec
5:	radial_velocity	km/s	barycentric radial velocity

The units used by this function are the units used in the `gaia_source` table.

This convenience function just invokes the 7-argument `astromUVW` function using the inverted parallax for the radial distance, and without invoking the Doppler correction. It is exactly equivalent to:

```
astromUVW(a[0], a[1], a[3], a[4], a[5], 1000./a[2], false)
```

Note this naive inversion of parallax to estimate distance is not in general reliable for parallaxes with non-negligable errors.

- Parameters:
  - `astrom6` *(array of floating point)*: vector of 6 astrometric parameters as provided by the Gaia source catalogue
- Return value
  - *(array of floating point)*: 3-element vector giving equatorial velocity components in km/s
- Examples:
  - `astromUVW(array(ra, dec, parallax, pmra, pmdec, radial_velocity))`
  - `icrsToGal(astromUVW(array(ra, dec, parallax, pmra, pmdec, radial_velocity)))`

`astromUVW( ra, dec, pmra, pmdec, radial_velocity, r_parsec, useDoppler )`

Calculates Cartesian components of velocity from the observed position and proper motion, radial velocity and radial distance, with optional light-time correction. The output is in the same coordinate system as the inputs, that is ICRS for the correspondingly-named Gaia quantities.

The radial distance must be supplied using the `r_parsec` parameter. A naive estimate from quantities in the Gaia source catalogue may be made with the expression `1000./parallax`, though note that this simple inversion of parallax is not in general reliable for parallaxes with non-negligable errors.

The calculations are fairly straightforward, following Eq. 1.5.74 from the Hipparcos catalogue. A (usually small) Doppler factor accounting for light-time effects can also optionally be applied. The effect of this is to multiply the returned vector by a factor of

`1/(1-radial_velocity/c)`, as discussed in Eq. 1.2.21 of the Hipparcos catalogue.

Note that no attempt is made to adjust for solar motion.

- Parameters:
  - `ra` *(floating point)*: Right Ascension in degrees
  - `dec` *(floating point)*: Declination in degrees
  - `pmra` *(floating point)*: proper motion in RA * cos(dec) in mas/yr
  - `pmdec` *(floating point)*: proper motion in declination in mas/yr
  - `radial_velocity` *(floating point)*: radial velocity in km/s
  - `r_parsec` *(floating point)*: radial distance in parsec
  - `useDoppler` *(boolean)*: whether to apply the Doppler factor to account for light-time effects
- Return value
  - *(array of floating point)*: 3-element vector giving equatorial velocity components in km/s
- Examples:
  - `astromUVW(ra, dec, pmra, pmdec, radial_velocity, dist, true)`
  - `icrsToGal(astromUVW(ra, dec, pmra, pmdec, radial_velocity, 1000./parallax, false))`

`epochProp( tYr, astrom6 )`

Propagates the astrometry parameters, supplied as a 6-element array, to a different epoch.

The input and output astrometry parameters are each represented by a 6-element array, with the following elements:

```
 index  gaia_source name  unit    description
 -----  ----------------  ----    -----------
   0:   ra                deg     right ascension
   1:   dec               deg     declination
   2:   parallax          mas     parallax
   3:   pmra              mas/yr  proper motion in ra * cos(dec)
   4:   pmdec             mas/yr  proper motion in dec
   5:   radial_velocity   km/s    barycentric radial velocity
```

The units used by this function are the units used in the `gaia_source` table.

- Parameters:
  - `tYr` *(floating point)*: epoch difference in years
  - `astrom6` *(array of floating point)*: astrometry at time t0, represented by a 6-element array as above (a 5-element array is also permitted where radial velocity is zero or unknown)
- Return value
  - *(array of floating point)*: astrometry at time `t0+tYr`, represented by a 6-element array as above
- Example:
  - `epochProp(-15.5, array(ra,dec,parallax,pmra,pmdec,radial_velocity))` - calculates the astrometry at 2000.0 of gaia_source values that were observed at 2015.5

`epochPropErr( tYr, astrom22 )`

Propagates the astrometry parameters and their associated errors and correlations, supplied as a

22-element array, to a different epoch.

The input and output astrometry parameters with associated error and correlation information are each represented by a 22-element array, with the following elements:

```
 index  gaia_source name      unit    description
 -----  ----------------      ----    -----------
   0:   ra                    deg     right ascension
   1:   dec                   deg     declination
   2:   parallax              mas     parallax
   3:   pmra                  mas/yr  proper motion in RA * cos(dec)
   4:   pmdec                 mas/yr  proper motion in Declination
   5:   radial_velocity       km/s    barycentric radial velocity
   6:   ra_error              mas     error in right ascension
   7:   dec_error             mas     error in declination
   8:   parallax_error        mas     error in parallax
   9:   pmra_error            mas/yr  error in RA proper motion * cos(dec)
  10:   pmdec_error           mas/yr  error in Declination proper motion
  11:   radial_velocity_error km/s    error in barycentric radial velocity
  12:   ra_dec_corr                   correlation between ra and dec
  13:   ra_parallax_corr              correlation between ra and parallax
  14:   ra_pmra_corr                  correlation between ra and pmra
  15:   ra_pmdec_corr                 correlation between ra and pmdec
  16:   dec_parallax_corr             correlation between dec and parallax
  17:   dec_pmra_corr                 correlation between dec and pmra
  18:   dec_pmdec_corr                correlation between dec and pmdec
  19:   parallax_pmra_corr            correlation between parallax and pmra
  20:   parallax_pmdec_corr           correlation between parallax and pmdec
  21:   pmra_pmdec_corr               correlation between pmra and pmdec
```

Note the correlation coefficients, always in the range -1..1, are dimensionless.

This is clearly an unwieldy function to invoke, but if you are using it with the gaia_source catalogue itself, or other similar catalogues with the same column names and units, you can invoke it by just copying and pasting the example shown in this documentation.

This transformation is only applicable for radial velocities determined independently of the astrometry, such as those obtained with a spectrometer. It is not applicable for the back-transformation of data already propagated to another epoch.

- Parameters:
  - `tYr` *(floating point)*: epoch difference in years
  - `astrom22` *(array of floating point)*: astrometry at time t0, represented by a 22-element array as above
- Return value
  - *(array of floating point)*: astrometry at time t0+tYr, represented by a 22-element array as above
- Example:
  - `epochPropErr(-15.5, array(ra,dec,parallax,pmra,pmdec,radial_velocity, ra_error,dec_error,parallax_error,pmra_error,pmdec_error,radial_velocity_error, ra_dec_corr,ra_parallax_corr,ra_pmra_corr,ra_pmdec_corr, dec_parallax_corr,dec_pmra_corr,dec_pmdec_corr, parallax_pmra_corr,parallax_pmdec_corr, pmra_pmdec_corr))` - calculates the astrometry with all errors and correlations at 2000.0 for gaia_source values that were observed at 2015.5.

`rvMasyrToKms( rvMasyr, plxMas )`

Converts from normalised radial velocity in mas/year to unnormalised radial velocity in km/s.

The output is calculated as `AU_YRKMS * rvMasyr / plxMas`, where `AU_YRKMS=4.740470446` is one Astronomical Unit in km.yr/sec.

- Parameters:
  - `rvMasyr` *(floating point)*: normalised radial velocity, in mas/year
  - `plxMas` *(floating point)*: parallax in mas
- Return value
  - *(floating point)*: radial velocity in km/s

`rvKmsToMasyr( rvKms, plxMas )`

Converts from unnormalised radial velocity in km/s to normalised radial velocity in mas/year.

The output is calculated as `rvKms * plxMas / AU_YRKMS`, where `AU_YRKMS=4.740470446` is one Astronomical Unit in km.yr/sec.

- Parameters:
  - `rvKms` *(floating point)*: unnormalised radial velocity, in mas/year
  - `plxMas` *(floating point)*: parallax in mas
- Return value
  - *(floating point)*: radial velocity in mas/year

`distanceEstimateEdsd( plxMas, plxErrorMas, lPc )`

Best estimate of distance using the Exponentially Decreasing Space Density prior. This estimate is provided by the mode of the PDF.

- Parameters:
  - `plxMas` *(floating point)*: parallax in mas
  - `plxErrorMas` *(floating point)*: parallax error in mas
  - `lPc` *(floating point)*: length scale in parsec
- Return value
  - *(floating point)*: best distance estimate in parsec

`distanceBoundsEdsd( plxMas, plxErrorMas, lPc )`

Calculates the 5th and 95th percentile confidence intervals on the distance estimate using the Exponentially Decreasing Space Density prior.

Note this function has to numerically integrate the PDF to determine quantile values, so it is relatively slow.

- Parameters:
  - `plxMas` *(floating point)*: parallax in mas
  - `plxErrorMas` *(floating point)*: parallax error in mas
  - `lPc` *(floating point)*: length scale in parsec
- Return value
  - *(array of floating point)*: 2-element array giving the 5th and 95th percentiles in parsec of the EDSD distance PDF

`distanceQuantilesEdsd( plxMas, plxErrorMas, lPc, qpoints, ... )`

Calculates arbitrary quantiles for the distance estimate using the Exponentially Decreasing Space Density prior.

Note this function has to numerically integrate the PDF to determine quantile values, so it is relatively slow.

- Parameters:
  - `plxMas` *(floating point)*: parallax in mas
  - `plxErrorMas` *(floating point)*: parallax error in mas
  - `lPc` *(floating point)*: length scale in parsec
  - `qpoints` *(floating point, one or more)*: one or more required quantile cut points, each in the range 0..1
- Return value
  - *(array of floating point)*: array with one element for each of the supplied `qpoints` giving the corresponding distance in parsec
- Examples:
  - `distanceQuantilesEdsd(parallax, parallax_error, 1350, 0.5)[0]` calculates the median of the EDSD distance PDF using a length scale of 1.35kpc
  - `distanceQuantilesEdsd(parallax, parallax_error, 3000, 0.01, 0.99)` returns a 2-element array giving the 1st and 99th percentile of the distance estimate using a length scale of 3kpc

`distanceToModulus( distPc )`
Converts a distance in parsec to a distance modulus. The formula is `5*log10(distPc)-5`.

- Parameters:
  - `distPc` *(floating point)*: distance in parsec
- Return value
  - *(floating point)*: distance modulus in magnitudes

`modulusToDistance( distmod )`
Converts a distance modulus to a distance in parsec. The formula is `10^(1+distmod/5)`.

- Parameters:
  - `distmod` *(floating point)*: distance modulus in magnitudes
- Return value
  - *(floating point)*: distance in parsec

`AU_YRKMS`
This quantity is A_v, the Astronomical Unit expressed in km.yr/sec. See the Hipparcos catalogue (ESA SP-1200) table 1.2.2 and Eq. 1.5.24.

`PC_AU`
Parsec in Astronomical Units, equal to 648000/PI.

`PC_YRKMS`
Parsec in units of km.yr/sec.

`C_KMS`
The speed of light in km/s (exact).

10.6.12 KCorrections

Functions for calculating K-corrections.

`kCorr( filter, redshift, colorType, colorValue )`

Calculates K-corrections. This allows you to determine K-corrections for a galaxy, given its redshift and a colour. Filters for GALEX, SDSS, UKIDSS, Johnson, Cousins and 2MASS are covered.

To define the calculation you must choose both a filter, specified as a `KCF_*` constant, and a colour (filter pair) specified as a `KCC_*` constant. For each available filter, only certain colours are available, as described in the documentation of the relevant `KCF_*` constant.

The algorithm used is described at http://kcor.sai.msu.ru/ (http://kcor.sai.msu.ru/). This is based on the paper *"Analytical Approximations of K-corrections in Optical and Near-Infrared Bands"* by I.Chilingarian, A.-L.Melchior and I.Zolotukhin (2010MNRAS.405.1409C (http://adsabs.harvard.edu/abs/2010MNRAS.405.1409C)), but extended to include GALEX UV bands and with redshift coverage up to 0.5 as described in *"Universal UV-optical Colour-Colour-Magnitude Relation of Galaxies"* by I.Chilingarian and I.Zolotukhin (2012MNRAS.419.1727C (http://adsabs.harvard.edu/abs/2012MNRAS.419.1727C)).

- Parameters:
  - `filter` *(KCorrections.KFilter)*: `KCF_*` constant defining the filter for which you want to calculate the K-correction
  - `redshift` *(floating point)*: galaxy redshift; this should be in the range 0-0.5
  - `colorType` *(KCorrections.KColor)*: `KCC_*` constant defining the filter pair for the calculation; check the `KCF_*` constant documentation to see which ones are permitted for a given filter
  - `colorValue` *(floating point)*: the value of the colour
- Return value
  - *(floating point)*: K correction
- Examples:
  - `kCorr(KCF_g, 0.16, KCC_gr, -0.8) = 3.593`
  - `kCorr(KCF_FUV, 0.48, KCC_FUVu, 0.31) = -0.170`

`KCF_FUV`

GALEX FUV filter (AB). Use with KCC_FUVNUV or KCC_FUVu.

`KCF_NUV`

GALEX NUV filter (AB). Use with KCC_NUVg or KCC_NUVr.

`KCF_u`

SDSS u filter (AB). Use with KCC_ur, KCC_ui or KCC_uz.

`KCF_g`

SDSS g filter (AB). Use with KCC_gr, KCC_gi or KCC_gz.

`KCF_r`

SDSS r filter (AB). Use with KCC_gr or KCC_ur.

KCF_i
SDSS i filter (AB). Use with KCC_gi or KCC_ui.

KCF_z
SDSS z filter (AB). Use with KCC_rz, KCC_gz or KCC_uz.

KCF_Y
UKIDSS Y filter (AB). Use with KCC_YH or KCC_YK.

KCF_J
UKIDSS J filter (AB). Use with KCC_JK or KCC_JH.

KCF_H
UKIDSS H filter (AB). Use with KCC_HK or KCC_JH.

KCF_K
UKIDSS K filter (AB). Use with KCC_JK or KCC_HK.

KCF_U
Johnson U filter (Vega). Use with KCC_URc.

KCF_B
Johnson B filter (Vega). Use with KCC_BRc or KCC_BIc.

KCF_V
Johnson V filter (Vega). Use with KCC_VIc or KCC_VRc.

KCF_Rc
Cousins Rc filter (Vega). Use with KCC_BRc or KCC_VRc.

KCF_Ic
Cousins Ic filter (Vega). Use with KCC_VIc.

KCF_J2
2MASS J filter (Vega). Use with KCC_J2Ks2 or KCC_J2H2.

KCF_H2
2MASS H filter (Vega). Use with KCC_H2Ks2 or KCC_J2H2.

KCF_Ks2
2MASS Ks filter (Vega). Use with KCC_J2Ks2 or KCC_H2Ks2.

KCC_BIc
Johnson B - Cousins Ic colour.

KCC_BRc
Johnson B - Cousins Rc colour.

KCC_FUVNUV
GALEX FUV - NUV colour.

KCC_FUVu
GALEX FUV - SDSS u colour.

KCC_gi
SDSS g - i colour.

KCC_gr
SDSS g - r colour.

KCC_gz
SDSS g - z colour.

KCC_H2Ks2
2MASS H - Ks colour.

KCC_HK
UKIDSS H - K colour.

KCC_J2H2
2MASS J - H colour.

KCC_J2Ks2
2MASS J - Ks colour.

KCC_JH
UKIDSS J - H colour.

KCC_JK
UKIDSS J - K colour.

KCC_NUVg
GALEX NUV - SDSS g colour.

KCC_NUVr
GALEX NUV - SDSS r colour.

KCC_rz
SDSS r - SDSS z colour.

KCC_ui
SDSS u - SDSS i colour.

KCC_URc
Johnson U - Cousins Rc colour.

KCC_ur
SDSS u - r colour.

KCC_uz
SDSS u - z colour.

KCC_VIc
Johnson V - Cousins Ic colour.

KCC_VRc
Johnson V - Cousins Rc colour.

KCC_YH
UKIDSS Y - H colour.

KCC_YK
UKIDSS Y - K colour.

10.6.13 Lists

Functions which operate on lists of values.

Some of these resemble similar functions in the `Arrays` class, and in some cases are interchangeable, but these are easier to use on non-array values because you don't have to explicitly wrap up lists of arguments as an array. However, for implementation reasons, most of the functions defined here can be used on values which are already `double[]` arrays (for instance array-valued columns) rather than as comma-separated lists of floating point values.

sum(values, ...)
Returns the sum of all the non-blank supplied arguments.

- Parameters:
  - `values` *(floating point, one or more)*: one or more numeric values

- Return value
  - *(floating point)*: sum of `values`
- Examples:
  - `sum(1, 3, 99) = 103`
  - `sum(1, 3, NaN) = 4`

`mean( values, ... )`
Returns the mean of all the non-blank supplied arguments.

- Parameters:
  - `values` *(floating point, one or more)*: one or more numeric values
- Return value
  - *(floating point)*: mean of `values`
- Examples:
  - `mean(2, 4, 6, 8) = 5`
  - `mean(100.5, 99.5, NaN) = 100`

`variance( values, ... )`
Returns the population variance of the non-blank supplied arguments.

- Parameters:
  - `values` *(floating point, one or more)*: one or more numeric values
- Return value
  - *(floating point)*: population variance of `values`
- Examples:
  - `variance(0, 3, 4, 3, 0) = 2.8`
  - `variance(0, 3, NaN, 3, NaN) = 2`

`stdev( values, ... )`
Returns the population standard deviation of the non-blank supplied arguments.

- Parameters:
  - `values` *(floating point, one or more)*: one or more numeric values
- Return value
  - *(floating point)*: population standard deviation of `values`
- Example:
  - `stdev(-3, -2, 0, 0, 1, 2, 3, 4, 5, 6) = 2.8`

`min( values, ... )`
Returns the minimum of all the non-blank supplied arguments.

- Parameters:
  - `values` *(floating point, one or more)*: one or more numeric values
- Return value

    - *(floating point)*: minimum of `values`
- Example:
    - `min(20, 25, -50, NaN, 101) = -50`

`max( values, ... )`
Returns the maximum of all the non-blank supplied arguments.

- Parameters:
    - `values` *(floating point, one or more)*: one or more numeric values
- Return value
    - *(floating point)*: maximum of `values`
- Example:
    - `max(20, 25, -50, NaN, 101) = 101`

`median( values, ... )`
Returns the median of all the non-blank supplied arguments.

- Parameters:
    - `values` *(floating point, one or more)*: one or more numeric values
- Return value
    - *(floating point)*: median of `values`
- Example:
    - `median(-100000, 5, 6, 7, 8) = 6`

`countTrue( values, ... )`
Returns the number of true values in a list of boolean arguments. Note if any of the values are blank, the result may be blank as well.

- Parameters:
    - `values` *(boolean, one or more)*: one or more true/false values
- Return value
    - *(integer)*: number of elements of `values` that are true
- Example:
    - `countTrue(false, false, true, false, true) = 2`

10.6.14 Maths

Standard mathematical and trigonometric functions. Trigonometric functions work with angles in radians.

`sin( theta )`
Sine of an angle.

- Parameters:

- `theta` *(floating point)*: an angle, in radians.

- Return value
  - *(floating point)*: the sine of the argument.

`cos( theta )`
Cosine of an angle.

- Parameters:
  - `theta` *(floating point)*: an angle, in radians.
- Return value
  - *(floating point)*: the cosine of the argument.

`tan( theta )`
Tangent of an angle.

- Parameters:
  - `theta` *(floating point)*: an angle, in radians.
- Return value
  - *(floating point)*: the tangent of the argument.

`asin( x )`
Arc sine of an angle. The result is in the range of -*pi*/2 through *pi*/2.

- Parameters:
  - x *(floating point)*: the value whose arc sine is to be returned.
- Return value
  - *(floating point)*: the arc sine of the argument (radians)

`acos( x )`
Arc cosine of an angle. The result is in the range of 0.0 through *pi*.

- Parameters:
  - x *(floating point)*: the value whose arc cosine is to be returned.
- Return value
  - *(floating point)*: the arc cosine of the argument (radians)

`atan( x )`
Arc tangent of an angle. The result is in the range of -*pi*/2 through *pi*/2.

- Parameters:
  - x *(floating point)*: the value whose arc tangent is to be returned.
- Return value
  - *(floating point)*: the arc tangent of the argument (radians)

`ln( x )`
Natural logarithm.

- Parameters:
  - x *(floating point)*: argument
- Return value
  - *(floating point)*: $\log_e(x)$

`exp( x )`
Euler's number *e* raised to a power.

- Parameters:
  - x *(floating point)*: the exponent to raise *e* to.
- Return value
  - *(floating point)*: the value e^x, where *e* is the base of the natural logarithms.

`log10( x )`
Logarithm to base 10.

- Parameters:
  - x *(floating point)*: argument
- Return value
  - *(floating point)*: $\log_{10}(x)$

`exp10( x )`
Power of 10. This convenience function is identical to `pow(10,x)`.

- Parameters:
  - x *(floating point)*: argument
- Return value
  - *(floating point)*: 10^x

`sqrt( x )`
Square root. The result is correctly rounded and positive.

- Parameters:
  - x *(floating point)*: a value.
- Return value
  - *(floating point)*: the positive square root of `x`. If the argument is NaN or less than zero, the result is NaN.

`square( x )`
Raise to the power 2.

- Parameters:
  - x *(floating point)*: a value
- Return value
  - *(floating point)*: x * x

`hypot( xs, ... )`
Returns the square root of the sum of squares of its arguments. In the 2-argument case, doing it like this may avoid intermediate overflow or underflow.

- Parameters:
  - `xs` *(floating point, one or more)*: one or more numeric values
- Return value
  - *(floating point)*: sqare root of sum of squares of arguments
- Examples:
  - `hypot(3,4) = 5`
  - `hypot(2,2,2,2) = 4`

`atan2( y, x )`
Converts rectangular coordinates (`x`,`y`) to polar (`r`,`theta`). This method computes the phase `theta` by computing an arc tangent of `y/x` in the range of -*pi* to *pi*.

- Parameters:
  - `y` *(floating point)*: the ordinate coordinate
  - `x` *(floating point)*: the abscissa coordinate
- Return value
  - *(floating point)*: the `theta` component (radians) of the point (`r`,`theta`) in polar coordinates that corresponds to the point (`x`,`y`) in Cartesian coordinates.

`pow( a, b )`
Exponentiation. The result is the value of the first argument raised to the power of the second argument.

- Parameters:
  - `a` *(floating point)*: the base.
  - `b` *(floating point)*: the exponent.
- Return value
  - *(floating point)*: the value a^b .

`sinh( x )`
Hyperbolic sine.

- Parameters:
  - `x` *(floating point)*: parameter
- Return value
  - *(floating point)*: result

`cosh( x )`
Hyperbolic cosine.

- Parameters:
  - `x` *(floating point)*: parameter

- Return value
  - *(floating point)*: result

tanh(x)
Hyperbolic tangent.

- Parameters:
  - x *(floating point)*: parameter
- Return value
  - *(floating point)*: result

asinh(x)
Inverse hyperbolic sine.

- Parameters:
  - x *(floating point)*: parameter
- Return value
  - *(floating point)*: result

acosh(x)
Inverse hyperbolic cosine.

- Parameters:
  - x *(floating point)*: parameter
- Return value
  - *(floating point)*: result

atanh(x)
Inverse hyperbolic tangent.

- Parameters:
  - x *(floating point)*: parameter
- Return value
  - *(floating point)*: result

E
Euler's number *e*, the base of natural logarithms.

PI
Pi, the ratio of the circumference of a circle to its diameter.

Infinity
Positive infinite floating point value.

NaN

Not-a-Number floating point value. Use with care; arithmetic and logical operations behave in strange ways near NaN (for instance, `NaN!=NaN`). For most purposes this is equivalent to the blank value.

10.6.15 Randoms

Functions concerned with random number generation.

There are two flavours of functions here: index-based (`random*`) and sequential (`nextRandom*`). Briefly, the index-based ones are safer to use, but provide poorer random statistics, while the sequential ones provide decent randomness but are not suitable for use in some/most contexts. They are documented separately below.

Index-based functions

The functions named `random*` all take an `index` parameter which determines the value of the result; the same index always leads to the same output, but there is not supposed to be any obvious relationship between index and output. An explicit index is required to ensure that a given cell always has the same value, since cell values are in general calculated on demand. The quality of the randomness for these functions may not be that good.

In most cases, the table row index, available as the special token `$0`, is a suitable value for the `index` parameter.

If several different random values are required in the same table row, one way is to supply a different row-based index value for each one, e.g. `random(2*$0)` and `random(2*$0+1)`. However, this tends to introduce a correlation between the random values in the same row, so a better (though in some cases slower) solution is to use one of the array-generating functions, e.g. `randomArray($0,2)[0]` and `randomArray($0,2)[1]`.

The output is deterministic, in the sense that the same invocation will always generate the same "random" number, even across different machines. However, in view of the comments in the implementation note below, the output *may* be subject to change in the future if some improved algorithm can be found, so this guarantee does not necessarily hold across software versions.

Implementation Note: The requirement for mapping a given input index deterministically to a pseudo-random number constrains the way that the random number generation is done; most well-studied RNGs generate sequences of random numbers, but that approach cannot be used here, since these sequences do not admit of random-access. What we do instead is to scramble the input index somewhat and use that as the seed for an instance of Java's `Random` class, which is then used to produce one or more random numbers per input index. Some thought and experimentation has gone into the current implementation (I bought a copy of Knuth Vol. 2 specially!) and an eyeball check of the results doesn't look all that bad, but it's still probably not very good, and is not likely to pass random number quality tests (though I haven't tried). A more respectable approach *might* be to use a cryptographic-grade hash function on the supplied index, but that's likely to be much slower. If there is demand, something like that could be added as an alternative option. In the mean time, beware if you use these random numbers for scientifically sensitive output.

Sequential functions

The functions named `nextRandom*` have no arguments, and supply the next value in a global sequence when they are evaluated. These can be used if scanning through a table once (for instance when writing a table using STILTS), but they are not suitable for contexts that should supply a

fixed value. For instance if you use them to define the value of a table cell in TOPCAT, that cell may have a different value every time you look at it, which may have disconcerting results. These use the java.util.Random class in a more standard way than the index-based functions and should provide random numbers of reasonable quality.

`random( index )`
Generates a pseudo-random number sampled from a uniform distribution between 0 and 1.

Note: The randomness may not be very high quality.

- Parameters:
  - `index` *(long integer)*: input value, typically row index "`$0`"
- Return value
  - *(floating point)*: random number between 0 and 1

`randomGaussian( index )`
Generates a pseudo-random number sampled from a Gaussian distribution with mean of 0.0 and standard deviation of 1.0.

Note: The randomness may not be very high quality.

- Parameters:
  - `index` *(long integer)*: input value, typically row index "`$0`"
- Return value
  - *(floating point)*: random number

`randomArray( index, n )`
Generates an array of pseudo-random numbers sampled from a uniform distribution between 0 and 1.

Note: The randomness may not be very high quality.

- Parameters:
  - `index` *(long integer)*: input value, typically row index "`$0`"
  - `n` *(integer)*: size of output array
- Return value
  - *(array of floating point)*: `n`-element array of random numbers between 0 and 1

`randomGaussianArray( index, n )`
Generates an array of pseudo-random numbers sampled from a Gaussian distribution with mean of 0.0 and standard deviation of 1.0.

Note: The randomness may not be very high quality.

- Parameters:
  - `index` *(long integer)*: input value, typically row index "`$0`"
  - `n` *(integer)*: size of output array
- Return value
  - *(array of floating point)*: `n`-element array of random numbers

`nextRandom( )`
Returns the next value in a random sequence, sampled from a uniform distribution between 0 and 1.

This function will give a different result every time, hence it is *not* suitable for use in an expression which should have a fixed value, for instance to define a TOPCAT column.

- Return value
  - *(floating point)*: random number between 0 and 1

`nextRandomGaussian( )`
Returns the next value in a random sequence, sampled from a Gaussian distribution with mean of 0.0 and standard deviation of 1.0.

This function will give a different result every time, hence it is *not* suitable for use in an expression which should have a fixed value, for instance to define a TOPCAT column.

- Return value
  - *(floating point)*: random number

10.6.16 Shapes

Functions useful for working with shapes in the (X, Y) plane.

`polyLine( x, xys, ... )`
Function of `x` defined by straight line segments between a specified list of vertices. The vertices are specified as a sequence of X_i, Y_i pairs, for which the X_i values must be monotonic. The line segment at each end of the specified point sequence is considered to be extended to infinity. If only two points are specified, this is the equation of a straight line between those points. As a special case, if only one point is specified, the line is considered to be a horizontal line (equal to the sole specified Y_i coordinate for all `x`).

By reversing the X_i and Y_i values, this function can equally be used to represent a function `X(y)` rather than `Y(x)`.

If the number of coordinates is odd, or the X_i values are not monotonic, behaviour is undefined.

- Parameters:
  - `x` *(floating point)*: X value at which function is to be evaluated
  - `xys` *(floating point, one or more)*: 2N arguments (`x1`, `y1`, `x2`, `y2`, ..., `xN`, `yN`) giving vertices of an N-point line with monotonically increasing or decreasing X values
- Return value
  - *(floating point)*: Y coordinate of poly-line for specified `x`
- Example:
  - `polyLine(5, 0,0, 2,2) = 5`

`isInside( x, y, xys, ... )`
Indicates whether a given test point is inside a polygon defined by specified list of vertices. The vertices are specified as a sequence of X_i, Y_i pairs.

If the number of coordinates is odd, the behaviour is not defined.

- Parameters:
  - `x` *(floating point)*: X coordinate of test point
  - `y` *(floating point)*: Y coordinate of test point
  - `xys` *(floating point, one or more)*: 2N arguments (`x1`, `y1`, `x2`, `y2`, ..., `xN`, `yN`) giving vertices of an N-sided polygon
- Return value
  - *(boolean)*: true iff test point is inside, or on the border of, the polygon
- Examples:
  - `isInside(0.5,0.5, 0,0, 0,1, 1,1, 1,0) = true`
  - `isInside(0,0, array(10,20, 20,20, 20,10)) = false`

10.6.17 Sky

Functions useful for working with shapes on a sphere. All angles are expressed in degrees.

`skyDistance( lon1, lat1, lon2, lat2 )`

Calculates the separation (distance around a great circle) of two points on the sky in degrees.

This function is identical to `skyDistanceDegrees` in class `CoordsDegrees`.

- Parameters:
  - `lon1` *(floating point)*: point 1 longitude in degrees
  - `lat1` *(floating point)*: point 1 latitude in degrees
  - `lon2` *(floating point)*: point 2 longitude in degrees
  - `lat2` *(floating point)*: point 2 latitude in degrees
- Return value
  - *(floating point)*: angular distance between point 1 and point 2 in degrees

`midLon( lon1, lon2 )`

Determines the longitude mid-way between two given longitudes. In most cases this is just the mean of the two values, but this function copes correctly with the case where the given values straddle the lon=0 line.

- Parameters:
  - `lon1` *(floating point)*: first longitude in degrees
  - `lon2` *(floating point)*: second longitude in degrees
- Return value
  - *(floating point)*: longitude midway between the given values
- Examples:
  - `midLon(204.0, 203.5) = 203.75`
  - `midLon(2, 359) = 0.5`

`midLat( lat1, lat2 )`

Determines the latitude midway between two given latitudes. This simply returns the mean of the two values, but is supplied for convenience to use alongside the `midLon` function.

- Parameters:
  - `lat1` *(floating point)*: first latitude in degrees
  - `lat2` *(floating point)*: second latitude in degrees
- Return value
  - *(floating point)*: latitude midway between the given values
- Example:
  - `midLat(23.5, 24.0) = 23.75`

`inSkyPolygon( lon0, lat0, lonLats, ... )`

Tests whether a given sky position is inside the polygon defined by a given set of vertices. The bounding lines of the polygon are the minor arcs of great circles between adjacent vertices, with an extra line assumed between the first and last supplied vertex. The interior of the polygon is defined to be the smaller of the two regions separated by the boundary. The vertices are specified as a sequence of lon_i, lat_i pairs.

The implementation of this point-in-polygon function is mostly correct, but may not be bulletproof. It ought to work for relatively small regions anywhere on the sky, but for instance it may get the sense wrong for regions that extend to cover both poles.

- Parameters:
  - `lon0` *(floating point)*: test point latitude in degrees
  - `lat0` *(floating point)*: test point longitude in degrees
  - `lonLats` *(floating point, one or more)*: 2N arguments (`lon1`, `lat1`, `lon2`, `lat2`, ..., `lonN`, `latN`) giving (longitude, latitude) vertices of an N-sided polygon in degrees
- Return value
  - *(boolean)*: true iff test point is inside, or on the border of, the polygon

10.6.18 Strings

String manipulation and query functions.

`concat( strings, ... )`

Concatenates multiple values into a string. In some cases the same effect can be achieved by writing `s1+s2+...`, but this method makes sure that values are converted to strings, with the blank value invisible.

- Parameters:
  - `strings` *(Object, one or more)*: one or more strings
- Return value
  - *(String)*: concatenation of input strings, without separators
- Examples:
  - `concat("blue", "moon") = "bluemoon"`
  - `concat("1", 2, 3, "4") = "1234"`
  - `concat("Astro", null, "Physics") = "AstroPhysics"`

`join( separator, words, ... )`

Joins multiple values into a string, with a given separator between each pair.

- Parameters:
  - `separator` *(String)*: string to insert between adjacent words
  - `words` *(Object, one or more)*: one or more values to join
- Return value
  - *(String)*: input values joined together with `separator`
- Examples:
  - `join("<->", "alpha", "beta", "gamma") = "alpha<->beta<->gamma"`
  - `join(" ", 1, "brown", "mouse") = "1 brown mouse"`

`equals( s1, s2 )`
Determines whether two strings are equal. Note you should use this function instead of `s1==s2`, which can (for technical reasons) return false even if the strings are the same.

- Parameters:
  - `s1` *(String)*: first string
  - `s2` *(String)*: second string
- Return value
  - *(boolean)*: true if s1 and s2 are both blank, or have the same content

`equalsIgnoreCase( s1, s2 )`
Determines whether two strings are equal apart from possible upper/lower case distinctions.

- Parameters:
  - `s1` *(String)*: first string
  - `s2` *(String)*: second string
- Return value
  - *(boolean)*: true if s1 and s2 are both blank, or have the same content apart from case folding
- Examples:
  - `equalsIgnoreCase("Cygnus", "CYGNUS") = true`
  - `equalsIgnoreCase("Cygnus", "Andromeda") = false`

`startsWith( whole, start )`
Determines whether a string starts with a certain substring.

- Parameters:
  - `whole` *(String)*: the string to test
  - `start` *(String)*: the sequence that may appear at the start of `whole`
- Return value
  - *(boolean)*: true if the first few characters of `whole` are the same as `start`
- Example:
  - `startsWith("CYGNUS X-1", "CYG") = true`

`endsWith( whole, end )`
Determines whether a string ends with a certain substring.

- Parameters:
  - `whole` *(String)*: the string to test
  - `end` *(String)*: the sequence that may appear at the end of `whole`
- Return value
  - *(boolean)*: true if the last few characters of `whole` are the same as `end`
- Example:
  - `endsWith("M32", "32") = true`

`contains( whole, sub )`
Determines whether a string contains a given substring.

- Parameters:
  - `whole` *(String)*: the string to test
  - `sub` *(String)*: the sequence that may appear within `whole`
- Return value
  - *(boolean)*: true if the sequence `sub` appears within `whole`
- Example:
  - `contains("Vizier", "izi") = true`

`length( str )`
Returns the length of a string in characters.

- Parameters:
  - `str` *(String)*: string
- Return value
  - *(integer)*: number of characters in `str`
- Example:
  - `length("M34") = 3`

`split( words )`
Splits a string into an array of space-separated words. One or more spaces separates each word from the next. Leading and trailing spaces are ignored.

The result is an array of strings, and if you want to use the individual elements you need to use square-bracket indexing, with `[0]` representing the first object

- Parameters:
  - `words` *(String)*: string with embedded spaces delimiting the words
- Return value
  - *(array of String)*: array of the separate words; you can extract the individual words from the result using square bracket indexing
- Examples:
  - `split("211:54:01 +29:33:41")` gives a 2-element array, first element is `"211:54:01"` and second element is `"+29:33:41"`.
  - `split(" cat dog cow ")[1] = "dog"`

`split( words, regex )`
Splits a string into an array of words separated by a given regular expression.

The result is an array of strings, and if you want to use the individual elements you need to use square-bracket indexing, with `[0]` representing the first object

- Parameters:
  - `words` *(String)*: string with multiple parts
  - `regex` *(String)*: regular expression delimiting the different words in the `words` parameter
- Return value
  - *(array of String)*: array of the separate words; you can extract the individual words from the result using square bracket indexing
- Examples:
  - `split("cat, dog, cow", ", *")` gives a 3-element string array.
  - `split("23.0, 45.92", ", ")[0] = "23.0"`
  - `parseDouble(split("23.0, 45.92", ", ")[0]) = 23`

`matches( str, regex )`
Tests whether a string matches a given regular expression.

- Parameters:
  - `str` *(String)*: string to test
  - `regex` *(String)*: regular expression string
- Return value
  - *(boolean)*: true if `regex` matches `str` anywhere
- Example:
  - `matches("Hubble", "ub") = true`

`matchGroup( str, regex )`
Returns the first grouped expression matched in a string defined by a regular expression. A grouped expression is one enclosed in parentheses.

- Parameters:
  - `str` *(String)*: string to match against
  - `regex` *(String)*: regular expression containing a grouped section
- Return value
  - *(String)*: contents of the matched group (or null, if `regex` didn't match `str`)
- Example:
  - `matchGroup("NGC28948b","NGC([0-9]*)") = "28948"`

`replaceFirst( str, regex, replacement )`
Replaces the first occurrence of a regular expression in a string with a different substring value.

- Parameters:
  - `str` *(String)*: string to manipulate

- `regex` *(String)*: regular expression to match in `str`
- `replacement` *(String)*: replacement string

- Return value
  - *(String)*: same as `str`, but with the first match (if any) of `regex` replaced by `replacement`
- Example:
  - `replaceFirst("Messier 61", "Messier ", "M-") = "M-61"`

`replaceAll( str, regex, replacement )`

Replaces all occurrences of a regular expression in a string with a different substring value.

- Parameters:
  - `str` *(String)*: string to manipulate
  - `regex` *(String)*: regular expression to match in `str`
  - `replacement` *(String)*: replacement string
- Return value
  - *(String)*: same as `str`, but with all matches of `regex` replaced by `replacement`
- Example:
  - `replaceAll("1-2--3---4","--*","x") = "1x2x3x4"`

`substring( str, startIndex )`

Returns the last part of a given string. The substring begins with the character at the specified index and extends to the end of this string.

- Parameters:
  - `str` *(String)*: the input string
  - `startIndex` *(integer)*: the beginning index, inclusive
- Return value
  - *(String)*: last part of `str`, omitting the first `startIndex` characters
- Example:
  - `substring("Galaxy", 2) = "laxy"`

`substring( str, startIndex, endIndex )`

Returns a substring of a given string. The substring begins with the character at `startIndex` and continues to the character at index `endIndex-1` Thus the length of the substring is `endIndex-startIndex`.

- Parameters:
  - `str` *(String)*: the input string
  - `startIndex` *(integer)*: the beginning index, inclusive
  - `endIndex` *(integer)*: the end index, inclusive
- Return value
  - *(String)*: substring of `str`
- Example:
  - `substring("Galaxy", 2, 5) = "lax"`

`toUpperCase( str )`
Returns an uppercased version of a string.

- Parameters:
  - `str` *(String)*: input string
- Return value
  - *(String)*: uppercased version of `str`
- Example:
  - `toUpperCase("Universe") = "UNIVERSE"`

`toLowerCase( str )`
Returns an lowercased version of a string.

- Parameters:
  - `str` *(String)*: input string
- Return value
  - *(String)*: lowercased version of `str`
- Example:
  - `toLowerCase("Universe") = "universe"`

`trim( str )`
Trims whitespace from both ends of a string.

- Parameters:
  - `str` *(String)*: input string
- Return value
  - *(String)*: str with any spaces trimmed from start and finish
- Examples:
  - `trim(" some text ") = "some text"`
  - `trim("some text") = "some text"`

`padWithZeros( value, ndigit )`
Takes an integer argument and returns a string representing the same numeric value but padded with leading zeros to a specified length.

- Parameters:
  - `value` *(long integer)*: numeric value to pad
  - `ndigit` *(integer)*: the number of digits in the resulting string
- Return value
  - *(String)*: a string evaluating to the same as `value` with at least `ndigit` characters
- Example:
  - `padWithZeros(23,5) = "00023"`

`desigToRa( designation )`

Attempts to determine the ICRS Right Ascension from an IAU-style designation such as "2MASS J04355524+1630331" following the specifications in the document https://cdsweb.u-strasbg.fr/Dic/iau-spec.html (https://cdsweb.u-strasbg.fr/Dic/iau-spec.html).

Note: this function should be used with considerable care. Such designators are intended for object identification and not for communicating sky positions, so that the resulting positions are likely to lack precision, and may be inaccurate. If positional information is available from other sources, it should almost certainly be used instead. But if there's no other choice, this may be used as a fallback.

Note also that a designator with no coordsystem-specific flag character (a leading "J", "B" or "G") is considered to be B1950, *not* J2000.

- Parameters:
  - `designation` *(String)*: designation string in IAU format
- Return value
  - *(floating point)*: ICRS right ascension in degreees, or blank if no position can be decoded
- Examples:
  - `desigToRa("2MASS J04355524+1630331") = 60.98016`
  - `desigToRa("PSR J120000.0+450000.0") = 180`
  - `desigToDec("PSR B120000.0+450000.0") = 180.639096`
  - `desigToRa("PN G001.2-00.3") = 267.403`
  - `desigToRa("NGC 4993") = NaN`

`desigToDec( designation )`

Attempts to determine the ICRS Declination from an IAU-style designation such as "2MASS J04355524+1630331" following the specifications in the document https://cdsweb.u-strasbg.fr/Dic/iau-spec.html (https://cdsweb.u-strasbg.fr/Dic/iau-spec.html).

Note: this function should be used with considerable care. Such designators are intended for object identification and not for communicating sky positions, so that the resulting positions are likely to lack precision, and may be inaccurate. If positional information is available from other sources, it should almost certainly be used instead. But if there's no other choice, this may be used as a fallback.

Note also that a designator with no coordsystem-specific flag character (a leading "J", "B" or "G") is considered to be B1950, *not* J2000.

- Parameters:
  - `designation` *(String)*: designation string in IAU format
- Return value
  - *(floating point)*: ICRS declination in degrees, or blank if no position can be decoded
- Examples:
  - `desigToDec("2MASS J04355524+1630331") = 16.50919`
  - `desigToDec("PSR J120000.0+450000.0") = 45`
  - `desigToDec("PSR B120000.0+450000.0") = 44.72167`
  - `desigToDec("PN G001.2-00.3") = -28.06457`
  - `desigToDec("NGC 4993") = NaN`

`desigToIcrs( designation )`

Attempts to decode an IAU-style designation such as "2MASS J04355524+1630331" to

determine its sky position, following the specifications in the document https://cdsweb.u-strasbg.fr/Dic/iau-spec.html (https://cdsweb.u-strasbg.fr/Dic/iau-spec.html).

Obviously, this only works where the *sequence* part of the designation takes one of the family of coordinate-based forms.

Note: this function should be used with considerable care. Such designators are intended for object identification and not for communicating sky positions, so that the resulting positions are likely to lack precision, and may be inaccurate. If positional information is available from other sources, it should almost certainly be used instead. But if there's no other choice, this may be used as a fallback.

Note also that a designator with no coordsystem-specific flag character (a leading "`J`", "`B`" or "`G`") is considered to be B1950, *not* J2000.

- Parameters:
  - `designation` *(String)*: designation string in IAU format
- Return value
  - *(array of floating point)*: 2-element array giving ICRS (RA,Dec) in degrees, or `null` if no position can be decoded

10.6.19 Tilings

Pixel tiling functions for the celestial sphere.

The `k` parameter for the HEALPix functions is the HEALPix order, which can be in the range 0<=k<=29. This is the logarithm to base 2 of the HEALPix NSIDE parameter. At order `k`, there are 12*4^k pixels on the sphere.

`htmIndex( level, lon, lat )`
Gives the HTM (Hierachical Triangular Mesh) pixel index for a given sky position.

- Parameters:
  - `level` *(integer)*: HTM level
  - `lon` *(floating point)*: longitude in degrees
  - `lat` *(floating point)*: latitude in degrees
- Return value
  - *(long integer)*: pixel index
- See Also:
  - HTM web site

`healpixNestIndex( k, lon, lat )`
Gives the pixel index for a given sky position in the HEALPix NEST scheme.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
  - `lon` *(floating point)*: longitude in degrees
  - `lat` *(floating point)*: latitude in degrees
- Return value

- *(long integer)*: pixel index

- See Also:
  - HEALPix web site

`healpixRingIndex( k, lon, lat )`

Gives the pixel index for a given sky position in the HEALPix RING scheme.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
  - `lon` *(floating point)*: longitude in degrees
  - `lat` *(floating point)*: latitude in degrees
- Return value
  - *(long integer)*: pixel index
- See Also:
  - HEALPix web site

`healpixNestLon( k, index )`

Returns the longitude of the approximate center of the tile with a given index in the HEALPix NEST scheme.

Note: the `index` parameter is 0-based, unlike the table row index special token `$index` (a.k.a. `$0`), which is 1-based. So if the HEALpix index is implicitly determined by the table row, the value `$index-1` should be used.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
  - `index` *(long integer)*: healpix index
- Return value
  - *(floating point)*: longitude in degrees
- See Also:
  - HEALPix web site

`healpixNestLat( k, index )`

Returns the latitude of the approximate center of the tile with a given index in the HEALPix NEST scheme.

Note: the `index` parameter is 0-based, unlike the table row index special token `$index` (a.k.a. `$0`), which is 1-based. So if the HEALpix index is implicitly determined by the table row, the value `$index-1` should be used.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
  - `index` *(long integer)*: healpix index
- Return value
  - *(floating point)*: latitude in degrees
- See Also:
  - HEALPix web site

`healpixRingLon( k, index )`

Returns the longitude of the approximate center of the tile with a given index in the HEALPix RING scheme.

Note: the `index` parameter is 0-based, unlike the table row index special token `$index` (a.k.a. `$0`), which is 1-based. So if the HEALpix index is implicitly determined by the table row, the value `$index-1` should be used.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
  - `index` *(long integer)*: healpix index
- Return value
  - *(floating point)*: longitude in degrees
- See Also:
  - HEALPix web site

`healpixRingLat( k, index )`

Returns the latitude of the approximate center of the tile with a given index in the HEALPix RING scheme.

Note: the `index` parameter is 0-based, unlike the table row index special token `$index` (a.k.a. `$0`), which is 1-based. So if the HEALpix index is implicitly determined by the table row, the value `$index-1` should be used.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
  - `index` *(long integer)*: healpix index
- Return value
  - *(floating point)*: latitude in degrees
- See Also:
  - HEALPix web site

`healpixNestToRing( k, nestIndex )`

Converts a healpix tile index from the NEST to the RING scheme at a given order.

Note: the `nestIndex` parameter is 0-based, unlike the table row index special token `$index` (a.k.a. `$0`), which is 1-based. So if the HEALpix index is implicitly determined by the table row, the value `$index-1` should be used.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
  - `nestIndex` *(long integer)*: pixel index in NEST scheme
- Return value
  - *(long integer)*: pixel index in RING scheme
- See Also:
  - HEALPix web site

`healpixRingToNest( k, ringIndex )`

Converts a healpix tile index from the RING to the NEST scheme at a given order.

Note: the `ringIndex` parameter is 0-based, unlike the table row index special token `$index` (a.k.a. `$0`), which is 1-based. So if the HEALpix index is implicitly determined by the table row, the value `$index-1` should be used.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
  - `ringIndex` *(long integer)*: pixel index in RING scheme
- Return value
  - *(long integer)*: pixel index in NEST scheme
- See Also:
  - HEALPix web site

`healpixK( pixelsize )`

Gives the HEALPix resolution parameter suitable for a given pixel size. This `k` value (also variously known as order, level, depth) is the logarithm to base 2 of the Nside parameter.

- Parameters:
  - `pixelsize` *(floating point)*: pixel size in degrees
- Return value
  - *(integer)*: HEALPix order `k`
- See Also:
  - HEALPix web site

`healpixResolution( k )`

Gives the approximate resolution in degrees for a given HEALPix resolution parameter `k` This `k` value is the logarithm to base 2 of the Nside parameter.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
- Return value
  - *(floating point)*: approximate angular resolution in degrees

`healpixSteradians( k )`

Returns the solid angle in steradians of each HEALPix pixel at a given order.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
- Return value
  - *(floating point)*: pixel size in steradians
- Examples:
  - `healpixSteradians(5) = 0.0010226538585904274`
  - `4*PI/healpixSteradians(0) = 12.0`

`healpixSqdeg( k )`

Returns the solid angle in square degrees of each HEALPix pixel at a given order.

- Parameters:
  - `k` *(integer)*: HEALPix order (0..29)
- Return value
  - *(floating point)*: pixel size in steradians
- Examples:
  - `healpixSqdeg(5) = 3.357174580844667`
  - `round(12 * healpixSqdeg(0)) = 41253`

`steradiansToSqdeg( sr )`

Converts a solid angle from steradians to square degrees.

The unit sphere is 4*PI steradians = 360*360/PI square degrees.

- Parameters:
  - `sr` *(floating point)*: quantity in steradians
- Return value
  - *(floating point)*: quantity in sqare degrees
- Example:
  - `round(steradiansToSqdeg(4*PI)) = 41253`

`sqdegToSteradians( sqdeg )`

Converts a solid angle from square degrees to steradians.

The unit sphere is 4*PI steradians = 360*360/PI square degrees.

- Parameters:
  - `sqdeg` *(floating point)*: quantity in square degrees
- Return value
  - *(floating point)*: quantity in steradians
- Example:
  - `round(sqdegToSteradians(41253)/PI) = 4`

`htmLevel( pixelsize )`

Gives the HTM `level` parameter suitable for a given pixel size.

- Parameters:
  - `pixelsize` *(floating point)*: required resolution in degrees
- Return value
  - *(integer)*: HTM level parameter

`htmResolution( level )`

Gives the approximate resolution in degrees for a given HTM depth level.

- Parameters:
  - `level` *(integer)*: HTM depth

- Return value
  - *(floating point)*: approximate angular resolution in degrees

SQDEG
Solid angle in steradians corresponding to 1 square degree.

10.6.20 Times

Functions for conversion of time values between various forms. The forms used are

Modified Julian Date (MJD)
A continuous measure in days since midnight at the start of 17 November 1858. Based on UTC.

Julian Day (JD)
MJD plus a fixed offset of 2400000.5. The number of days since the notional creation of the universe, midday on 1 Jan 4713 BC.

ISO 8601
A string representation of the form `yyyy-mm-ddThh:mm:ss.s`, where the `T` is a literal character (a space character may be used instead). Based on UTC.

Julian Epoch
A continuous measure based on a Julian year of exactly 365.25 days. For approximate purposes this resembles the fractional number of years AD represented by the date. Sometimes (but not here) represented by prefixing a 'J'; J2000.0 is defined as 2000 January 1.5 in the TT timescale.

Besselian Epoch
A continuous measure based on a tropical year of about 365.2422 days. For approximate purposes this resembles the fractional number of years AD represented by the date. Sometimes (but not here) represented by prefixing a 'B'.

Decimal Year
Fractional number of years AD represented by the date. 2000.0, or equivalently 1999.99recurring, is midnight at the start of the first of January 2000. Because of leap years, the size of a unit depends on what year it is in.

Therefore midday on the 25th of October 2004 is `2004-10-25T12:00:00` in ISO 8601 format, 53303.5 as an MJD value, 2004.81588 as a Julian Epoch and 2004.81726 as a Besselian Epoch.

Currently this implementation cannot be relied upon to better than a millisecond.

isoToMjd(isoDate)
Converts an ISO8601 date string to Modified Julian Date. The basic format of the `isoDate` argument is `yyyy-mm-ddThh:mm:ss.s`, though some deviations from this form are permitted:

- The '`T`' which separates date from time can be replaced by a space
- The seconds, minutes and/or hours can be omitted
- The decimal part of the seconds can be any length, and is optional
- A '`Z`' (which indicates UTC) may be appended to the time

Some legal examples are therefore: "`1994-12-21T14:18:23.2`", "`1968-01-14`", and "`2112-05-25 16:45Z`".

- Parameters:
  - `isoDate` *(String)*: date in ISO 8601 format
- Return value
  - *(floating point)*: modified Julian date corresponding to `isoDate`
- Examples:
  - `isoToMjd("2004-10-25T18:00:00") = 53303.75`
  - `isoToMjd("1970-01-01") = 40587.0`

`dateToMjd( year, month, day, hour, min, sec )`
Converts a calendar date and time to Modified Julian Date.

- Parameters:
  - `year` *(integer)*: year AD
  - `month` *(integer)*: index of month; January is 1, December is 12
  - `day` *(integer)*: day of month (the first day is 1)
  - `hour` *(integer)*: hour (0-23)
  - `min` *(integer)*: minute (0-59)
  - `sec` *(floating point)*: second (0<=sec<60)
- Return value
  - *(floating point)*: modified Julian date corresponding to arguments
- Example:
  - `dateToMjd(1999, 12, 31, 23, 59, 59.) = 51543.99998`

`dateToMjd( year, month, day )`
Converts a calendar date to Modified Julian Date.

- Parameters:
  - `year` *(integer)*: year AD
  - `month` *(integer)*: index of month; January is 1, December is 12
  - `day` *(integer)*: day of month (the first day is 1)
- Return value
  - *(floating point)*: modified Julian date corresponding to 00:00:00 of the date specified by the arguments
- Example:
  - `dateToMjd(1999, 12, 31) = 51543.0`

`decYearToMjd( decYear )`
Converts a Decimal Year to a Modified Julian Date.

- Parameters:
  - `decYear` *(floating point)*: decimal year
- Return value
  - *(floating point)*: modified Julian Date
- Example:
  - `decYearToMjd(2000.0) = 51544.0`

`isoToUnixSec( isoDate )`

Converts an ISO8601 date string to seconds since 1970-01-01. The basic format of the `isoDate` argument is `yyyy-mm-ddThh:mm:ss.s`, though some deviations from this form are permitted:

- The '`T`' which separates date from time can be replaced by a space
- The seconds, minutes and/or hours can be omitted
- The decimal part of the seconds can be any length, and is optional
- A '`Z`' (which indicates UTC) may be appended to the time

Some legal examples are therefore: "`1994-12-21T14:18:23.2`", "`1968-01-14`", and "`2112-05-25 16:45Z`".

- Parameters:
  - `isoDate` *(String)*: date in ISO 8601 format
- Return value
  - *(floating point)*: seconds since the Unix epoch
- Examples:
  - `isoToUnixSec("2004-10-25T18:00:00") = 1098727200`
  - `isoToMjd("1970-01-01") = 0`

`decYearToUnixSec( decYear )`

Converts a Decimal Year to seconds since 1970-01-01.

- Parameters:
  - `decYear` *(floating point)*: decimal year
- Return value
  - *(floating point)*: seconds since the Unix epoch
- Examples:
  - `decYearToUnixSec(2000.0) = 946684800`
  - `decYearToUnixSec(1970) = 0`

`mjdToUnixSec( mjd )`

Converts a Modified Julian Date to seconds since 1970-01-01.

- Parameters:
  - `mjd` *(floating point)*: modified Julian date
- Return value
  - *(floating point)*: seconds since the Unix epoch

`jdToUnixSec( jd )`

Converts a Julian day to seconds since 1970-01-01.

- Parameters:
  - `jd` *(floating point)*: Julian day
- Return value
  - *(floating point)*: seconds since the Unix epoch

`mjdToIso( mjd )`
Converts a Modified Julian Date value to an ISO 8601-format date-time string. The output format is `yyyy-mm-ddThh:mm:ss`. If the result predates the Common Era, the string "(BCE)" is prepended.

- Parameters:
  - `mjd` *(floating point)*: modified Julian date
- Return value
  - *(String)*: ISO 8601 format date corresponding to `mjd`
- Example:
  - `mjdToIso(53551.72917) = "2005-06-30T17:30:00"`

`mjdToDate( mjd )`
Converts a Modified Julian Date value to an ISO 8601-format date string. The output format is `yyyy-mm-dd`. If the result predates the Common Era, the string "(BCE)" is prepended.

- Parameters:
  - `mjd` *(floating point)*: modified Julian date
- Return value
  - *(String)*: ISO 8601 format date corresponding to `mjd`
- Example:
  - `mjdToDate(53551.72917) = "2005-06-30"`

`mjdToTime( mjd )`
Converts a Modified Julian Date value to an ISO 8601-format time-only string. The output format is `hh:mm:ss`.

- Parameters:
  - `mjd` *(floating point)*: modified Julian date
- Return value
  - *(String)*: ISO 8601 format time corresponding to `mjd`
- Example:
  - `mjdToTime(53551.72917) = "17:30:00"`

`mjdToDecYear( mjd )`
Converts a Modified Julian Date to Decimal Year.

- Parameters:
  - `mjd` *(floating point)*: modified Julian Date
- Return value
  - *(floating point)*: decimal year
- Example:
  - `mjdToDecYear(0.0) = 1858.87671`

`formatMjd( mjd, format )`

Converts a Modified Julian Date value to a date using a customisable date format. The format is as defined by the `java.text.SimpleDateFormat` (http://docs.oracle.com/javase/6/docs/api/java/text/SimpleDateFormat.html) class. The default output corresponds to the string "`yyyy-MM-dd'T'HH:mm:ss`"

Note that the output from certain formatting characters (such as `MMM` for month, `EEE` for day of week) is dependent on your locale (system language settings). The output time zone however always corresponds to UTC.

- Parameters:
  - `mjd` *(floating point)*: modified Julian date
  - `format` *(String)*: formatting patttern
- Return value
  - *(String)*: custom formatted time corresponding to `mjd`
- Examples:
  - `formatMjd(50000.3, "EEE dd, MMM, yy") = "Tue 10 Oct, 95"`
  - `formatMjd(50000.1234, "'time 'H:mm:ss.SSS") = "time 2:57:41.760"`

`jdToMjd( jd )`

Converts a Julian Day to Modified Julian Date. The calculation is simply `jd-2400000.5`.

- Parameters:
  - `jd` *(floating point)*: Julian day number
- Return value
  - *(floating point)*: MJD value

`mjdToJd( mjd )`

Converts a Modified Julian Date to Julian Day. The calculation is simply `jd+2400000.5`.

- Parameters:
  - `mjd` *(floating point)*: MJD value
- Return value
  - *(floating point)*: Julian day number

`mjdToJulian( mjd )`

Converts a Modified Julian Date to Julian Epoch. For approximate purposes, the result of this routine consists of an integral part which gives the year AD and a fractional part which represents the distance through that year, so that for instance 2000.5 is approximately 1 July 2000.

- Parameters:
  - `mjd` *(floating point)*: modified Julian date
- Return value
  - *(floating point)*: Julian epoch
- Example:
  - `mjdToJulian(0.0) = 1858.87885`

`julianToMjd( julianEpoch )`
Converts a Julian Epoch to Modified Julian Date. For approximate purposes, the argument of this routine consists of an integral part which gives the year AD and a fractional part which represents the distance through that year, so that for instance 2000.5 is approximately 1 July 2000.

- Parameters:
  - `julianEpoch` *(floating point)*: Julian epoch
- Return value
  - *(floating point)*: modified Julian date
- Example:
  - `julianToMjd(2000.0) = 51544.5`

`mjdToBesselian( mjd )`
Converts Modified Julian Date to Besselian Epoch. For approximate purposes, the result of this routine consists of an integral part which gives the year AD and a fractional part which represents the distance through that year, so that for instance 1950.5 is approximately 1 July 1950.

- Parameters:
  - `mjd` *(floating point)*: modified Julian date
- Return value
  - *(floating point)*: Besselian epoch
- Example:
  - `mjdToBesselian(0.0) = 1858.87711`

`besselianToMjd( besselianEpoch )`
Converts Besselian Epoch to Modified Julian Date. For approximate purposes, the argument of this routine consists of an integral part which gives the year AD and a fractional part which represents the distance through that year, so that for instance 1950.5 is approximately 1 July 1950.

- Parameters:
  - `besselianEpoch` *(floating point)*: Besselian epoch
- Return value
  - *(floating point)*: modified Julian date
- Example:
  - `besselianToMjd(1950.0) = 33281.92346`

`unixMillisToMjd( unixMillis )`
Converts from milliseconds since the Unix epoch (1970-01-01T00:00:00) to a modified Julian date value

- Parameters:
  - `unixMillis` *(long integer)*: milliseconds since the Unix epoch
- Return value

- *(floating point)*: modified Julian date

mjdToUnixMillis(mjd)
Converts from modified Julian date to milliseconds since the Unix epoch (1970-01-01T00:00:00).

- Parameters:
  - `mjd` *(floating point)*: modified Julian date
- Return value
  - *(long integer)*: milliseconds since the Unix epoch

MJD_OFFSET
JD value for MJD = 0 (=2400000.5).

10.6.21 TrigDegrees

Standard trigonometric functions with angles in degrees.

sinDeg(theta)
Sine of an angle.

- Parameters:
  - `theta` *(floating point)*: an angle, in degrees
- Return value
  - *(floating point)*: the sine of the argument

cosDeg(theta)
Cosine of an angle.

- Parameters:
  - `theta` *(floating point)*: an angle, in degrees
- Return value
  - *(floating point)*: the cosine of the argument

tanDeg(theta)
Tangent of an angle.

- Parameters:
  - `theta` *(floating point)*: an angle, in degrees
- Return value
  - *(floating point)*: the tangent of the argument.

asinDeg(x)
Arc sine. The result is in the range of -90 through 90.

- Parameters:
  - x *(floating point)*: the value whose arc sine is to be returned.
- Return value
  - *(floating point)*: the arc sine of the argument in degrees

acosDeg(x)
Arc cosine. The result is in the range of 0.0 through 180.

- Parameters:
  - x *(floating point)*: the value whose arc cosine is to be returned.
- Return value
  - *(floating point)*: the arc cosine of the argument in degrees

atanDeg(x)
Arc tangent. The result is in the range of -90 through 90.

- Parameters:
  - x *(floating point)*: the value whose arc tangent is to be returned.
- Return value
  - *(floating point)*: the arc tangent of the argument in degrees

atan2Deg(y, x)
Converts rectangular coordinates (x,y) to polar (r,theta). This method computes the phase theta by computing an arc tangent of y/x in the range of -180 to 180.

- Parameters:
  - y *(floating point)*: the ordinate coordinate
  - x *(floating point)*: the abscissa coordinate
- Return value
  - *(floating point)*: the theta component in degrees of the point (r,theta) in polar coordinates that corresponds to the point (x,y) in Cartesian coordinates.

10.6.22 URLs

Functions that construct URLs for external services. Most of the functions here just do string manipulation to build up URL strings, using knowledge of the parameters required for various services.

urlEncode(txt)
Performs necessary quoting of a string for it to be included safely in the data part of a URL. Alphanumeric characters and the characters underscore ("_"), minus sign ("-"), period (".") and tilde ("~") are passed through unchanged, and any other 7-bit ASCII character is represented by a percent sign ("%") followed by its 2-digit hexadecimal code. Characters with values of 128 or greater are simply dropped.

- Parameters:

- `txt` *(String)*: input (unencoded) string

- Return value
  - *(String)*: output (encoded) string
- Example:
  - `urlEncode("RR Lyr") = "RR%20Lyr"`
- See Also:
  - RFC 3986

`urlDecode( txt )`

Reverses the quoting performed by `urlEncode`. Percent-encoded sequences (`%xx`) are replaced by the ASCII character with the hexadecimal code `xx`.

- Parameters:
  - `txt` *(String)*: input (encoded) string
- Return value
  - *(String)*: output (unencoded) string
- Example:
  - `urlDecode("RR%20Lyr") = "RR Lyr"`
- See Also:
  - RFC 3986

`paramsUrl( baseUrl, nameValuePairs, ... )`

Builds a query-type URL string given a base URL and a list of name, value pairs.

The parameters are encoded on the command line according to the "`application/x-www-form-urlencoded`" convention, which appends a "?" to the base URL, and then adds name=value pairs separated by "&" characters, with percent-encoding of non-URL-friendly characters. This format is used by many services that require a list of parameters to be conveyed on the URL.

- Parameters:
  - `baseUrl` *(String)*: basic URL (may or may not already contain a "?")
  - `nameValuePairs` *(String, one or more)*: an even number of arguments (or an even-length string array) giving parameter name1,value1,name2,value2,...nameN,valueN
- Return value
  - *(String)*: form-encoded URL
- Example:
  - `paramsUrl("http://x.org/", "a", "1", "b", "two", "c", "3&4") = "http://x.org/?a=1&b=two&c=3%264"`

`bibcodeUrl( bibcode )`

Maps a bibcode to the URL that will display the relevant entry in ADS (https://ui.adsabs.harvard.edu/). If the supplied string does not appear to be a bibcode, null will be returned.

If the supplied string appears to be a bibcode, it just prepends the string

`https://ui.adsabs.harvard.edu/abs/`" and performs any character escaping that is required.

- Parameters:
  - `bibcode` *(String)*: ADS-style bibcode string
- Return value
  - *(String)*: display URL pointing at bibcode record, or null if it doesn't look like a bibcode
- Example:
  - `bibcodeUrl("2018A&A...616A...2L") = "https://ui.adsabs.harvard.edu/abs/2018A%26A...616A...2L"`
- See Also:
  - http://adsabs.harvard.edu/abs_doc/help_pages/data.html

`doiUrl( doi )`

Maps a DOI (Digital Object Identifier) to its dislay URL. If the supplied string does not appear to be a DOI, null will be returned.

If the supplied string appears to be a DOI, it strips any "`doi:`" prefix if present, prepends the string "`https://doi.org/`", and performs any character escaping that is required.

- Parameters:
  - `doi` *(String)*: DOI string, with or without "doi:" prefix
- Return value
  - *(String)*: display URL pointing at DOI content, or null if it doesn't look like a DOI
- Example:
  - `doiUrl("10.3390/informatics4030018") = "https://doi.org/10.3390/informatics4030018"`
- See Also:
  - https://www.doi.org/

`arxivUrl( arxivId )`

Maps an arXiv identifier to the URL that will display its arXiv (https://arxiv.org/) web page. If the supplied string does not appear to be an arXiv identifier, null will be returned.

If the supplied string appears to be an arXiv identifier, it strips any "`arXiv:` prefix and prepends the string "`https://arxiv.org/abs/`".

- Parameters:
  - `arxivId` *(String)*: arXiv identifier, with or without "arXiv:" prefix
- Return value
  - *(String)*: display URL pointing at bibcode record, or null if it doesn't look like a bibcode
- Example:
  - `arxivUrl("arXiv:1804.09379") = "https://arxiv.org/abs/1804.09381"`
- See Also:
  - https://arxiv.org/help/arxiv_identifier

`simbadUrl( sourceId )`

Maps a source identifier to the URL of its SIMBAD (http://simbad.u-strasbg.fr/simbad/) web page. SIMBAD is the astronomical source information service run by the Centre de Données astronomiques de Strasbourg.

The string "`http://simbad.u-strasbg.fr/simbad/sim-id?Ident=`" is prepended to the given id string, and any necessary character escaping is applied. No attempt is made to validate whether the supplied string is a real source identifier, so there is no guarantee that the returned URL will contain actual results.

- Parameters:
  - `sourceId` *(String)*: free text assumed to represent a source identifier known by SIMBAD
- Return value
  - *(String)*: URL of the Simbad web page describing the identified source
- Example:
  - `simbadUrl("Beta Pictoris") = "http://simbad.u-strasbg.fr/simbad/sim-id?Ident=Beta%20Pictoris"`

`nedUrl( sourceId )`

Maps a source identifier to the URL of its NED (http://ned.ipac.caltech.edu/) web page. NED is the NASA/IPAC Extragalactic Database.

The string "`http://ned.ipac.caltech.edu/byname?objname=`" is prepended to the given id string, and any necessary character escaping is applied. No attempt is made to validate whether the supplied string is a real source identifier, so there is no guarantee that the returned URL will contain actual results.

- Parameters:
  - `sourceId` *(String)*: free text assumed to represent a source identifier known by NED
- Return value
  - *(String)*: URL of the NED web page describing the identified source
- Example:
  - `nedUrl("NGC 3952") = "http://ned.ipac.caltech.edu/byname?objname=NGC%203952"`

`hips2fitsUrl( hipsId, fmt, raDeg, decDeg, fovDeg, npix )`

Returns the URL of a cutout from the Hips2Fits service operated by CDS. The result will be the URL of a FITS or image file resampled to order from one of the HiPS surveys available at CDS.

This function requests a square cutout using the SIN projection, which is suitable for small cutouts. If the details of this function don't suit your purposes, you can construct the URL yourself.

- Parameters:
  - `hipsId` *(String)*: identifier or partial identifier for the HiPS survey
  - `fmt` *(String)*: required output format, for instance "`fits`", "`png`", "`jpg`"
  - `raDeg` *(floating point)*: central Right Ascension (longitude) in degrees
  - `decDeg` *(floating point)*: central Declination (latitude) in degrees

- `fovDeg` *(floating point)*: field of view; extent of the cutout in degrees
- `npix` *(integer)*: extent of the cutout in pixels (width=height=npix)

- Return value
  - *(String)*: URL of the required cutout
- See Also:
  - http://alasky.u-strasbg.fr/hips-image-services/hips2fits

10.6.23 VO

Virtual Observatory-specific functions. Some of these are for rather technical purposes.

The UCD parsing functions are based on Grégory Mantelet's library Ucidy (https://github.com/gmantele/ucidy) corresponding to UCD1+ 1.4 (https://www.ivoa.net/documents/UCD1+/20210616/), and the VOUnit parsing functions are based on Norman Gray's library Unity (http://www.astro.gla.ac.uk/users/norman/ivoa/unity/) corresponding to VOUnits 1.0 (https://www.ivoa.net/documents/VOUnits/20140523/).

`ucdStatus( ucd )`

Returns a token string giving the category of UCD compliance. UCD1+, UCD1 and SIAv1 ("VOX:")-style UCDs are recognised.

Possible return values are currently:

- "OK": conforms to UCD1+ standard
- "UCD1": looks like a UCD1
- "VOX": is in VOX: namespace introduced by SIAv1
- "BAD_SYNTAX": not a UCD1 and cannot be parsed according to UCD1+
- "BAD_SEQUENCE": UCD words violate UCD1+ sequence rules
- "UNKNOWN_WORD": follows UCD1+ syntax rules but contains non-UCD1+ atom
- "NAMESPACE": follows UCD1+ syntax but contains atoms in non-standard namespace
- "DEPRECATED": contains deprecated UCD1+ words

In the case of non-compliant values, more information can be found using the `ucdMessage` function.

- Parameters:
  - `ucd` *(String)*: UCD string
- Return value
  - *(String)*: "OK" for conformant UCD1+, otherwise some other value

`ucdMessage( ucd )`

Returns a human-readable message associated with the parsing of a UCD. This is expected to be empty for a fully compliant UCD, and may contain error messages or advice otherwise.

- Parameters:
  - `ucd` *(String)*: UCD value
- Return value
  - *(String)*: message text

`vounitStatus( unit )`
Returns a token string giving the category of VOUnits compliance.

Possible return values are currently:

- "OK": conforms to VOUnits syntax
- "UNKNOWN_UNIT": parsed as VOUnit but contains unknown base unit(s)
- "GUESSED_UNIT": parsed as VOUnit but contains unknown, though guessable, base unit(s)
- "BAD_SYNTAX": cannot be parsed as VOUnit syntax
- "PARSE_ERROR": unexpected error during parsing
- "USAGE": violates VOUnit usage constraints
- "WHITESPACE": legal VOUnit except that it contains whitespace, which is not allowed by VOUnits

In the case of non-compliant values, more information can be found using the `vounitMessage` function.

- Parameters:
  - `unit` *(String)*: unit string
- Return value
  - *(String)*: "OK" for conformant VOUnits, otherwise some other value

`vounitMessage( unit )`
Returns a human-readable message associated with the parsing of a unit string. This is expected to be empty for a string fully compliant with VOUnits, and may contain error messages or additional information otherwise.

- Parameters:
  - `unit` *(String)*: unit string
- Return value
  - *(String)*: message text

10.7 Examples

Here are some examples for defining new columns; the expressions below could appear as the `<expr>` in a `tpipe addcol` or `sortexpr` command).

Average

```
(first + second) * 0.5
```

Square root

```
sqrt(variance)
```

Angle conversion

```
radiansToDegrees(DEC_radians)
degreesToRadians(RA_degrees)
```

Conversion from string to number

```
parseInt($12)
parseDouble(ident)
```

Conversion from number to string

```
toString(index)
```

Conversion between numeric types

```
toShort(obs_type)
toDouble(range)
```

or

```
(short) obs_type
(double) range
```

Conversion from sexagesimal to degrees

```
hmsToDegrees(RA1950)
dmsToDegrees(decDeg,decMin,decSec)
```

Conversion from degrees to sexagesimal

```
degreesToDms($3)
degreesToHms(RA,2)
```

Outlier clipping

```
min(1000, max(value, 0))
```

Converting a magic value to null

```
jmag == 9999 ? NULL : jmag
```

Converting a null value to a magic one

```
NULL_jmag ? 9999 : jmag
```

Taking the third scalar element from an array-valued column

```
psfCounts[2]
```

Converting spectral type to numeric value (e.g. "L3.5" -> 23.5, "M7" -> 17)

```
"MLT".indexOf(spType.charAt(0)) * 10 + parseDouble(substring(spType,1)) + 10
```

Note this uses a couple of Java String instance methods (Section 10.8.2) which are not explicitly documented in this section.

Here are some examples of boolean expressions that could be used for row selection (appearing in a `tpipe select` command)

Within a numeric range

```
RA > 100 && RA < 120 && Dec > 75 && Dec < 85
```

Within a circle

```
$2*$2 + $3*$3 < 1
```

```
skyDistanceDegrees(ra0,dec0,hmsToDegrees(RA),dmsToDegrees(DEC))<15./3600.
```

First 100 rows

```
index <= 100
```

(though you could use `tpipe cmd='head 100'` instead)

Every tenth row

```
index % 10 == 0
```

(though you could use `tpipe cmd='every 10'` instead)

String equality/matching

```
equals(SECTOR, "ZZ9 Plural Z Alpha")
equalsIgnoreCase(SECTOR, "zz9 plural z alpha")
startsWith(SECTOR, "ZZ")
contains(ph_qual, "U")
```

String regular expression matching

```
matches(SECTOR, "[XYZ] Alpha")
```

Test for non-blank value

```
! NULL_ellipticity
```

10.8 Advanced Topics

This section contains some notes on getting the most out of the algebraic expressions facility. If you're not a Java programmer, some of the following may be a bit daunting - read on at your own risk!

10.8.1 Expression evaluation

This note provides a bit more detail for Java programmers on what is going on here; it describes how the use of functions in STILTS algebraic expressions relates to normal Java code.

The expressions which you write are compiled to Java bytecode when you enter them (if there is a 'compilation error' it will be reported straight away). The functions listed in the previous subsections are all the `public static` methods of the classes which are made available by default. The classes listed are all in the package `uk.ac.starlink.ttools.func`. However, the public static methods are all imported into an anonymous namespace for bytecode compilation, so that you write (`sqrt(x,y)` and not `Maths.sqrt(x,y)`. The same happens to other classes that are imported (which can be in any package or none) - their public static methods all go into the anonymous namespace. Thus, method name clashes are a possibility.

This cleverness is all made possible by the rather wonderful JEL (http://www.gnu.org/software/jel/).

10.8.2 Instance Methods

There is another category of functions which can be used apart from those listed in Section 10.6. These are called, in Java/object-oriented parlance, "instance methods" and represent functions that can be executed on an object.

It is possible to invoke any of its public instance methods on any object (though not on primitive values - numeric and boolean ones). The syntax is that you place a "." followed by the method invocation after the object you want to invoke the method on, hence `NAME.substring(3)` instead of `substring(NAME,3)`. If you know what you're doing, feel free to go ahead and do this. However, most of the instance methods you're likely to want to use have equivalents in the normal functions listed in the previous section, so unless you're a Java programmer or feeling adventurous, you may be best off ignoring this feature.

10.8.3 Adding User-Defined Functions

The functions provided by default for use with algebraic expressions, while powerful, may not provide all the operations you need. For this reason, it is possible to write your own extensions to the expression language. In this way you can specify abritrarily complicated functions. Note however that this will only allow you to define new columns or subsets where each cell is a function only of the other cells in the same row - it will not allow values in one row to be functions of values in another.

In order to do this, you have to write and compile a (probably short) program in the Java language. A full discussion of how to go about this is beyond the scope of this document, so if you are new to Java and/or programming you may need to find a friendly local programmer to assist (or mail the author). The following explanation is aimed at Java programmers, but may not be incomprehensible to non-specialists.

The steps you need to follow are:

1. Write and compile a class containing one or more static public methods representing the function(s) required
2. Make this class available on the application's classpath at runtime as described in Section 3.1
3. Specify the class's name to the application, as the value of the `jel.classes` system property (colon-separated if there are several) as described in Section 3.3

Any public static methods defined in the classes thus specified will then be available for use. They should be defined to take and return the relevant primitive or Object types for the function required. For instance a class written as follows would define a three-value average:

```
public class AuxFuncs {
    public static double average3(double x, double y, double z) {
        return (x + y + z) / 3.0;
    }
}
```

and the command

```
stilts tpipe cmd='addcol AVERAGE "average3($1,$2,$3)"'
```

would add a new column named AVERAGE giving the average of the first three existing columns. Exactly how you would build this is dependent on your system, but it might involve doing something like the following:

1. Writing a file named `AuxFuncs.java` containing the above code
2. Compiling it using a command like "`javac AuxFuncs.java`"
3. Running `tpipe` using the flags "`stilts -classpath . -Djel.classes=AuxFuncs tpipe`"

Note that (in versions later than STILTS 3.0-6) variable-length argument lists are supported where the final declared argument is an array, so for instance a method declared like:

```
public static double sum(double[] values) {
    double total = 0;
```

```
            for (int i = 0; i < values.length; i++) {
                total += values[i];
            }
            return total;
        }
```

can be invoked like "`sum(UMAG,GMAG,RMAG,IMAG,ZMAG)`". The alternative form "`double... values`" can be used in the declaration with identical effect.

11 Server Mode

STILTS is generally run by issuing commands from some kind of command line. However, facilities are also included to run it as a service, accepting commands via HTTP. The basic implementation of this is in the form of some Servlet implementations that can be run in a servlet container. At present the following servlets are provided:

- `PlotServlet`
- `TaskServlet`
- `FormServlet`

In order to run them there are a few options:

STILTS `server` command
: The simplest is to use the STILTS `server` command. If you run this, the services will start up immediately and print information about the base URL. Some configuration of available functions and data access is offered by the command line parameters. This command is a thin wrapper around the Jetty servlet engine which is included in the STILTS distribution.

Deploy in a servlet container
: For more control you can deploy the servlet or servlets you want to run in a servlet container of your choice, for instance Tomcat. The servlets can be configured by setting context parameters in a `web.xml` file in the usual way; look at the `StiltsContext` class to see the available configuration parameters.

Docker
: In the case of the plot server, a Docker container has been provided for convenience. At time of writing this is available, with some more discussion about how to deploy it, at https://hub.docker.com/r/mbtaylor/plotserv.

In all cases, look at the base URL of the running service for more information and usage examples. For instance if the server is running at the default base URL, then pointing your browser at

```
http://localhost:2112/stilts/
```

will show you links to documentation for the available services.

The different services are described further in the following subsections.

Note: The `server` command and associated servlet code are somewhat experimental. If you have requirements which are not currently provided, please contact the author for discussion.

11.1 Plot Service

The plot server supports interactive plots that can be viewed and navigated using HTTP requests. A plot is specified with a URL, corresponding to an invocation of one of the `plot2*` commands, that specifies a plot, and uses a service API that can be used by client-side javascript to display and update a plot image in response to user navigation actions such as mouse drag and wheel gestures. The effect is of an interactive plot in a web browser that can be navigated with the mouse in exactly the same way as a STILTS or TOPCAT interactive plot.

To see this in action, run

```
% stilts server
```

point your browser at the URL displayed and follow the links to see some interactive plots.

The following subsections give some more details about how this works and how it can be used.

11.1.1 Usage

The best way to see how the plot service can be used is to run the service (most simply by executing `stilts server`), and look at the running plot examples, along with their HTML source code, served from the server endpoint.

However this section gives a brief independent overview of how the service can be used. The PlotServlet itself exposes a RESTful API as described in Section 11.1.2. This offers the services required for client HTML/JavaScript code running in a browser to set up a plot based on server-side (or at least server-accessible) table data, and to make updates to it following user activity like mouse gestures.

A basic client JavaScript library named `plot2Lib.js`, that facilitates this plot setup is included with STILTS; it can be found in the `stilts.jar` file at `uk/ac/starlink/ttools/server/plot2Lib.js`, and is available from the running server at `<plot-service-base>/plot2Lib.js`. It exports some variables in the `plot2` namespace; see comments in the file itself for documentation of how to use it. A minimal web page using this to present an interactive plot might look like:

```
<html>
<script src="plot2Lib.js"></script>
<script>
   var serverUrl = "http://localhost:2112/stilts/plot/";
   onload = function() {
      var plotTxt = plot2.wordsToPlotTxt([
         "plot2sky",
         "layer1=mark",
         "in1=/data/catalogue.fits",
         "lon1=RA",
         "lat1=DEC"
      ]);
      var plotNode = plot2.createPlotNode(serverUrl, plotTxt);
      document.getElementsByTagName("body")[0].appendChild(plotNode);
   };
</script>
</html>
```

In this case the data file "`/data/catalogue.fits`" has to be available to the servlet; how this is arranged depends on the way the service is deployed.

JavaScript clients don't have to use the `plot2Lib.js` library; it's also possible to write custom code that talks to the RESTful API described in Section 11.1.2.

11.1.2 RESTful API

This section details the endpoints supported by the plot servlet; it is not required reading for those who want to use the supplied `plot2Lib.js` JavaScript library, but may be of interest to those who want to write their own JavaScript clients.

General URL Syntax

The plot service accepts URLs of the form

```
<base-url>/<action-type>/<plot-spec>[?<session-id>&<arg-list>]
```

These parts are expanded as follows:

`<action-type>`
: The action type string determines the kind of request, and is one of the strings "`html`", "`state`",

"imgsrc", "position", "count", "row"; see below for details.

<plot-spec>
The plot specification is an ampersand-separated STILTS plot command string; the form is "<command-name>&<arg-name>=<value>&<arg-name>=<value>..." for instance "plot2sky&layer1=mark&in1=stars.vot&lon1=ra&lat1=dec". See STILTS plotting documentation in Section 8 for command syntax. Note that although this part contains &-separated name=value pairs which are syntactically application/x-www-form-urlencoded it is part of the URI path and *not* a URI query string since it does not come after a question mark ('?') and it *cannot* be supplied by POSTing parameters.

<session-id>
The session identifier is of the form "sessionId=<unique-string>" and it serves to maintain the state of the plot between requests (so that for instance a navigation action starts from where the last one left off). The <unique-string> string is chosen by the client; any string value is permitted, but it's up to the client to pick something that is unlikely to be chosen by other unrelated clients on the same or different machines. Incorporating a high-resolution timestamp is a good bet. In case of a collision, confusing results may ensue, but it's probably not necessary to resort to cryptographic-grade hashing. This part is not necessary for the "html" <action-type>.

<arg-list>
A list of zero or more ampersand-separated "<name>=<value>" parameters, specific to the <action-type>; see below for details.

The [?<session-id>&<arg-list>] part of the URL is an application/x-www-form-urlencoded query-string, and may be supplied as a POST body rather than as part of the GET query if preferred. Note that does *not* apply to the <plot-spec> part, which is in RFC3986 terms part of the *path* and not part of the *query*.

Available Action Types

The options for the various different values of the <action-type> string, with their associated parameters and response values, are as follows:

html
Returns a new standalone HTML page containing the interactive plot specified by the supplied plot specification. Note this does not require a session ID to be supplied, and it has no parameters. This action is easy to use, but not very flexible.

state
Returns a JSON structure giving the image and/or image annotations for the current state of the session. The session state may be updated by supplying navigation parameters, as follows:

- navigate=reset: reset the plot to initial state
- navigate=resize&size=width,height: resize the plot to width x height pixels
- navigate=click&pos=x,y&ibutton=1|2|3: emulate TOPCAT click on plot at graphics position x,y
- navigate=drag&origin=x0,y0&pos=x1,y1&ibutton=1|2|3: emulate TOPCAT continuing drag from start graphics position x0,y0 to x1,y1
- navigate=drag&origin=x0,y0&pos=x1,y1&ibutton=1|2|3&end=true: emulate TOPCAT end-drag from start graphics position x0,y0 to x1,y1
- navigate=wheel&pos=x,y&wheelrot=nstep: emulate TOPCAT mouse wheel at graphics position x,y
- navigate=none: no change to last position

The members of the returned structure are:

`imgSrc` (present if plot has changed since last request):
Contains a `data:` URL suitable for use as the content of an `IMG/@src` attribute to display the current state of the plot

`staticSvg` (present if there are static decorations):
SVG content, suitable as the content of an `SVG` node, giving decorations to superimpose over the plot image.

`transientSvg` (present if there are transient decorations):
SVG content, suitable as the content of an `SVG` node, giving decorations to superimpose over the plot image representing a navigation action. Such decorations should only be displayed for a short time (e.g. 0.5 second).

`bounds` (present for planar and cubic plots):
A 2- or 3-element array of (lower,upper) data bound pairs giving the extent of the current plot.

`imgsrc`
Returns image data suitable for reference by an `IMG/@src` attribute value for the current state of the session. The session state may be updated by supplying navigation parameters as follows:

- `navigate=reset`: reset the plot to initial state
- `navigate=resize&size=width,height`: resize the plot to `width` x `height` pixels
- `navigate=click&pos=x,y&ibutton=1|2|3`: emulate TOPCAT click on plot at graphics position `x,y`
- `navigate=drag&origin=x0,y0&pos=x1,y1&ibutton=1|2|3`: emulate TOPCAT continuing drag from start graphics position `x0,y0` to `x1,y1`
- `navigate=drag&origin=x0,y0&pos=x1,y1&ibutton=1|2|3&end=true`: emulate TOPCAT end-drag from start graphics position `x0,y0` to `x1,y1`
- `navigate=wheel&pos=x,y&wheelrot=nstep`: emulate TOPCAT mouse wheel at graphics position `x,y`
- `navigate=none`: no change to last position

The response MIME type can be influenced by the `ofmt` parameter in the `<plot-spec>` *or* the `format` parameter in the `<arg-list>`. In case of disagreement, the latter takes precedence. The available options are "`png`", "`png-transp`", "`gif`", "`jpeg`", "`pdf`", "`svg`", "`eps`", "`eps-gzip`".

`position`
Provides a service to convert a position in plot graphics coordinates to data coordinates. The request must have a `pos` argument giving the graphics position in the form `x,y` (e.g. "`pos=203,144`"), and the response will be a JSON structure with the following members:

`dataPos`:
On success, contains data coordinates as a numeric array.

`txtPos`:
On success, contains data coordinates as a formatted string.

`message`:
Textual indication of conversion status. It will be "`ok`" on success, but may have some other value, such as "`out of plot bounds`", on failure.

`count`
Returns the number of points currently visible in this plot. Note that determining this value does take some computation (it's not free). The output is a plain text decimal value; it can also be interpreted as JSON.

`row`
Services a request for row information, at or near a submitted graphics position on the plot.

The request must have a `pos` argument giving the graphics position in the form `x,y` (e.g. "`pos=203,144`"),

The output is a JSON array of objects; there is one array element for each table represented, and each such element is a column-name->column-value map for the row indicated. If the submitted point is not near any plotted points, the result will therefore be an empty array.

This API is subject to extension or modification in future releases.

11.1.3 Caching and Performance

The plot servlet requires some temporary disk storage for caching images and prepared coordinate data to improve performance when updating plots during visualisation sequences. Servlet configuration options are available to manage usage of these resources.

When first requested to make a plot, it works out what coordinate data will be required, reads this from the input table, and writes it to cache in an efficient binary storage format. This may be simply a copy of columns from the input table, or may require some computation, for instance if the plotted quantity is the result of an expression in the expression language, or if the coordinates are longitude and latitude (in which case they are converted to unit vector components). This coordinate cache preparation requires a scan of the input table which is in the current implementation single-threaded, so may be time-consuming for very large tables.

Once the coordinate data is in cache, the cache rather than the input table is used for subsequent plots; that includes both derived replots resulting from user navigation and completely separate plots that happen to require the same row/column data. Plots from cache are in most cases multi-threaded, so can be quite rapid even for large datasets depending on server configuration and load.

When the plot is first made, the initial (pre-navigation) plotted image is also stored in a cache, so that if other HTTP clients request the same plot they can receive the initial image without further data access or computation.

By default, the directory used for caching is the value of the `java.io.tmpdir` system property, which is typically `/tmp` on Unix systems. The directory used for caching can be changed by modifying this system property, or caching can be configured with the servlet context parameters controlled by the `StiltsContext` class.

When cache storage gets full, older items are dropped in the usual way. There is no doubt scope for improvement of the existing cache management; the details may be refined in future releases.

11.2 Task Service

This servlet allows execution of a single STILTS command by making an HTTP request with the command name and parameters, rather than by presenting them on the command line. You might want to run this service if you are providing a web service to external users which is able to access files residing on the server, for instance generating static table plots (though see the Plot service for more sophisticated server-side visualisation) or row selections on the fly. This can be done without the server mode, for instance by invoking the `stilts` script or java from a CGI script to serve each request, but using server mode has two advantages: first it provides correct output HTTP headers such as Content-Type, and secondly it avoids the Java startup overheads for each invocation.

The URLs accepted by the service are of the form

```
<task-base-url>/<task-name>?<arg-name>=<arg-value>&<arg-name>=<arg-value>...
```

The <task-name> is one of the STILTS command names as listed in Appendix B, and the <arg-name>=<arg-value> pairs are the parameter settings for that command; the parameter settings are syntactically an application/x-www-form-urlencoded query string. These parameters may be supplied as a POST body rather than as part of a GET query if preferred, though since the task invocation will normally be idempotent, GET may be more respectable.

The HTTP response depends on the behaviour of the task in question. If the task is one that outputs a table or sequence of tables, such as tpipe or tmatch2, the HTTP response will be the serialized form of the table in an appropriate format. By default that is currently VOTable (Content-Type "application/x-votable+xml"), but that can be altered by supplying the appropriate output format parameter, e.g. "ofmt=fits". Plotting commands return an image in a default graphics format; again the output format can be controlled using command paramters in the same way as on the command line. Output for some other commands such as calc may just be plain text.

Here are some example invocations, assuming the default server base URL of http://localhost:2112/stilts/:

http://localhost:2112/stilts/

Returns an HTML page giving version information and some links to example usages of the server.

http://localhost:2112/stilts/task/tpipe

Returns an HTML page giving usage instructions for the tpipe task.

http://localhost:2112/stilts/task/calc?expression=21%2b2

Invokes the calc task to return a document containing the text "23". Note that the plus ("+") sign in the expression has to be encoded using the sequence "%2b" since "+" has a special significance in query URLs - see for instance sec 2.2 of RFC 1738.

http://localhost:2112/stilts/task/plot2plane?in=/data/table1.vot&layer1=mark&x1=RMAG&y1=BMAG

Invokes the plot2d task to return a magnitude-magnitude diagram of the named server-side file as an image, probably an image/png (but see the Plot Service if you want interactive plots).

http://localhost:2112/stilts/task/tcopy?in=/data/cat.fits&ofmt=ecsv

Invokes the tcopy task to return a translation of the named server-side FITS file to ECSV format.

Note that in the current form of this service no great attention has been paid to security, so it may be possible for clients to read and write files and expend significant system resources by making certain requests to the server. Anyone exposing this service directly to external clients should bear this in mind.

11.3 Form Service

The Form service is more or less obsolete; it really just gives an example of how to set up an HTML Form to invoke one of the STILTS commands using the Task Service.

The content served gives provides a couple of examples of an HTML form that can invoke plotting using the Task Service. The output graphics use the old-style plot commands rather than the much more capable plot2* plotting, and are not interactive; for much improved server-side visualisastion,

see Section 11.1.

12 Programmatic Invocation

The STILTS package provides some capabilities, for instance plotting, that might be useful as part of other Java applications. The code that forms STILTS is fully documented at the API level; there are comprehensive javadocs throughout for the `uk.ac.starlink.ttools` package, its subpackages, and most of the other classes in the `uk.ac.starlink` tree on which it relies. Anybody is welcome to use these classes at their own risk, but the code does not form a stable API intended for public use: the javadocs are not distributed as part of the package (though you may be able to find them here), tutorial documentation is not provided, and there is no commitment to API stability between releases.

With this in mind, there are facilities for invoking the STILTS commands programmatically from third-party java code. Of course it is possible to do this by just calling the static `main(String[])` method of the application Main-Class (`Stilts`) but we document here how it can be done in a way which allows more control, using the `uk.ac.starlink.task` parameter handling framework.

Each of the STILTS tasks listed in Appendix B is represented by a class implementing the `Task` interface; these all have no-arg constructors. To run it, you need to create an instance of the class, pass it an `Environment` object which can acquire values for parameters by name, and then execute it. The `MapEnvironment` class, based on a Map containing name/value pairs, is provided for this purpose. As well as managing parameter values, MapEnvironment captures table and text output in a way that lets you retrieve it after the task has executed. Here is a simple example for invoking the calc task to perform a simple calcation:

```
MapEnvironment env = new MapEnvironment();
env.setValue( "expression", "sqrt(3*3+4*4)" );
Task calcTask = new uk.ac.starlink.ttools.task.Calc();
calcTask.createExecutable( env ).execute();
String result = env.getOutputText();
```

The execution corresponds exactly to the command-line:

```
stilts calc expression="sqrt(3*3+4*4)"
```

The Usage section for the `calc` task notes that the corresponding Task subclass is `Calc`.

Also in the usage section, each parameter reports the data type that it may take, and objects of this type may be used as the parameter value passed in the `MapEnvironment` as an alternative to passing string values. For the case of the input table parameters, this is `StarTable`, so in a task like tpipe (`TablePipe`), if you want to read a file "data.fits", you can either write

```
env.setValue( "in", "data.fits" );
```

or

```
StarTable table = new StarTableFactory().readStarTable( "data.fits" );
env.setValue( "in", table );
```

That doesn't buy you much, but the table could equally be obtained from any other source, including being a user-defined iterable over existing data structures. See SUN/252 for more information on `StarTable` handling.

For some short examples of programs which invoke STILTS tasks in this way, see the source code of some of the examples in the `uk.ac.starlink.ttools.example` directory: Calculator and Head10.

Some commands provide additional methods for use with parameter-based invocation. In particular the plotting commands can be used to create JComponent objects that can be incorporated into an existing GUI. A working example of this can be found in the source code for the example

EnvPlanePlotter class. For some more tutorial introductions to using the plotting classes programmatically, see also the example classes SinePlot, ApiPlanePlotter, and BasicPlotGui in the same place.

A Commands By Category

This section lists the commands available broken down by the category of function they provide. Some commands appear in more than one category. Detailed descriptions and examples for each command can be found in Appendix B.

Format conversion:

- `tcopy` (Appendix B.29): Converts between table formats
- `votcopy` (Appendix B.42): Transforms between VOTable encodings

See also Section 5.

Generic table manipulation:

- `tcopy` (Appendix B.29): Converts between table formats
- `tpipe` (Appendix B.39): Performs pipeline processing on a table
- `tmulti` (Appendix B.37): Writes multiple tables to a single container file
- `tmultin` (Appendix B.38): Writes multiple processed tables to single container file
- `tcat` (Appendix B.27): Concatenates multiple similar tables
- `tcatn` (Appendix B.28): Concatenates multiple tables
- `tloop` (Appendix B.31): Generates a single-column table from a loop variable
- `tjoin` (Appendix B.33): Joins multiple tables side-to-side
- `arrayjoin` (Appendix B.1): Adds table-per-row data as array-valued columns
- `tgridmap` (Appendix B.32): Calculates N-dimensional density maps
- `tcube` (Appendix B.30): Calculates N-dimensional histograms

See also Section 6.

Crossmatching:

- `tmatch1` (Appendix B.34): Performs a crossmatch internal to a single table
- `tmatch2` (Appendix B.35): Crossmatches 2 tables using flexible criteria
- `tmatchn` (Appendix B.36): Crossmatches multiple tables using flexible criteria
- `tskymatch2` (Appendix B.41): Crossmatches 2 tables on sky position
- `cdsskymatch` (Appendix B.3): Crossmatches table on sky position against VizieR/SIMBAD table
- `coneskymatch` (Appendix B.5): Crossmatches table on sky position against remote cone service
- `sqlskymatch` (Appendix B.21): Crossmatches table on sky position against SQL table

See also Section 7.

Plotting:

- `plot2plane` (Appendix B.10): Draws a plane plot
- `plot2sky` (Appendix B.11): Draws a sky plot
- `plot2cube` (Appendix B.12): Draws a cube plot
- `plot2sphere` (Appendix B.13): Draws a sphere plot
- `plot2time` (Appendix B.14): Draws a time plot
- `plot2d` (Appendix B.15) *(deprecated)*: Old-style 2D Scatter Plot
- `plot3d` (Appendix B.16) *(deprecated)*: Old-style 3D Scatter Plot
- `plothist` (Appendix B.17) *(deprecated)*: Old-style Histogram

See also Section 9.

Sky Pixel Operations:

- `tskymap` (Appendix B.40): Calculates sky density maps
- `pixfoot` (Appendix B.8): Generates Multi-Order Coverage maps
- `pixsample` (Appendix B.9): Samples from a HEALPix pixel data file

VOTables:

- `votcopy` (Appendix B.42): Transforms between VOTable encodings
- `votlint` (Appendix B.43): Validates VOTable documents

Virtual Observatory service access:

- `cdsskymatch` (Appendix B.3): Crossmatches table on sky position against VizieR/SIMBAD table
- `cone` (Appendix B.4): Executes a Cone Search-like query
- `coneskymatch` (Appendix B.5): Crossmatches table on sky position against remote cone service
- `tapskymatch` (Appendix B.26): Crossmatches table on sky position against TAP table
- `tapquery` (Appendix B.24): Queries a Table Access Protocol server
- `tapresume` (Appendix B.25): Resumes a previous query to a Table Access Protocol server
- `taplint` (Appendix B.23): Tests TAP services
- `datalinklint` (Appendix B.6): Validates DataLink documents
- `regquery` (Appendix B.18): Queries the VO registry

SQL Database access:

- `sqlclient` (Appendix B.20): Executes SQL statements
- `sqlupdate` (Appendix B.22): Updates values in an SQL table
- `sqlskymatch` (Appendix B.21): Crossmatches table on sky position against SQL table

Miscellaneous:

- `server` (Appendix B.19): Runs an HTTP server to perform STILTS commands
- `calc` (Appendix B.2): Evaluates expressions
- `funcs` (Appendix B.7): Browses functions used by algebraic expression language
- `xsdvalidate` (Appendix B.44): Validates against XML Schema

B Command Reference

This appendix provides the reference documentation for the commands in the package. For each one a description of its purpose, a list of its command-line arguments, and some examples are given.

B.1 `arrayjoin`: Adds table-per-row data as array-valued columns

`arrayjoin` takes an input table and for each row adds the contents of a separate "array" table. The columns added are the columns from the array table, and the value of each cell is the value of the whole column from the array table represented as an array. The assumption is that all the array tables have the same form (the same columns, though not necessarily the same row counts).

This can be useful for constructing a single table with array-valued columns containing data that is made available in multiple external files, for instance via the DataLink protocol; this is illustrated in the Examples subsection below. Note however that this command does not understand DataLink directly, and cannot itself determine the location of the external array tables; an expression giving their per-row location (filname or URL) must be supplied.

B.1.1 Usage

The usage of `arrayjoin` is

```
stilts <stilts-flags> arrayjoin ifmt=<in-format> istream=true|false
                                icmd=<cmds> ocmd=<cmds>
                                omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp
                                out=<out-table> ofmt=<out-format>
                                atable=<loc-expr> afmt=<in-format>
                                astream=true|false acmd=<cmds>
                                cache=true|false fixcols=none|dups|all
                                suffixarray=<label>
                                [in=]<table>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.ArrayJoin`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`acmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on array tables as specified by parameter `atable`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`afmt = <in-format>` ***(String)***

Specifies the format of array tables as specified by parameter `atable`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program

will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`astream = true|false` ***(Boolean)***
If set true, array tables specified by the `atable` parameter will be read as a stream. It is necessary to give the `afmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`atable = <loc-expr>` ***(String)***
Gives the location of the table whose rows will be turned into an array-valued column. This will generally be an expression giving a URL or filename that is different for each row of the input table. If table loading fails for the given location, for instance becase the file is not found or an HTTP 404 response is received, the array cells in the corresponding row will be blank.

The first non-blank table loaded defines the array columns to be added. If subsequent tables have a different structure (do not contain similar columns in a similar sequence) an error may result. If the external array tables are not all homogenous in this way, the `acmd` parameter can be used to filter them so that they are.

`cache = true|false` ***(Boolean)***
Determines whether the array data will be cached the first time an array table is read (true) or re-read from the array table every time the row is accessed (false). Since the row construction may be an expensive step, especially if the tables are downloaded, it usually makes sense to set this true (the default). When true it also enables the metadata to be adjusted to report constant array length where applicable, which cannot be done before all the rows have been scanned, and which may enable more efficient file output. However, if you want to stream the data you can set it false.

[Default: `true`]

`fixcols = none|dups|all` ***(Fixer)***
Determines how input columns are renamed before use in the output table. The choices are:

- `none`: columns are not renamed
- `dups`: columns which would otherwise have duplicate names in the output will be renamed to indicate which table they came from
- `all`: all columns will be renamed to indicate which table they came from

If columns are renamed, the new ones are determined by `suffix*` parameters.

[Default: `dups`]

`icmd = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ifmt = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

in = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

istream = true|false ***(Boolean)***

If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

ocmd = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ofmt = <out-format> ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`
- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

`out = <out-table>` ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: -]

`suffixarray = <label>` ***(String)***

If the `fixcols` parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from the array tables.

[Default: `_a`]

B.1.2 Examples

Here are some examples of using `arrayjoin`:

```
stilts arrayjoin in=dr3-sources.vot
                 atable='"https://gea.esac.esa.int/data-server/data?RETRIEVAL_TYPE=XP_SAMPLE
                 icmd=progress
                 out=sources-with-xp.fits
```

This command uses the Gaia DR3 DataLink service to attach sampled XP spectrum data to an input table containing other information about Gaia DR3 sources.

The input file `dr3-sources.vot` must contain the column `source_id` giving the Gaia DR3 source ID corresponding to that row. The `atable` expression is a URL built using that `source_id` column and a base URL that can be obtained by examining the DR3 catalogue documentation, or by examining the DataLink service descriptor returned from a source catalogue query.

The output file `sources-with-xp.fits` is a FITS table with the same content as the input but with three added array-valued columns, `wavelength`, `flux` and `flux_error` as supplied by

tables downloaded from the URL in the `atable` parameter. For each row of the output table, the lengths of those three arrays will be the same, namely the row count of the table that supplied them. These matched arrays can be manipulated within STILTS, for instance using the Arrays functions or to produce plots using the lines or arrayquantile plot layer types.

The `icmd=progress` filter is a useful convenience to see how the command is progressing, since the operation requires multiple downloads and so may be time-consuming.

```
stilts arrayjoin in=dl_dr3.fits
                 icmd='select has_epoch_photometry'
                 atable='"https://gea.esac.esa.int/data-server/data?RETRIEVAL_TYPE=EPOCH_PHO
                 afmt=votable \
                 acmd='select equals(BAND,\"G\")' \
                 acmd='keepcols "time mag flux"' \
                 out=with-epoch-photom.fits
```

Similar to the previous example, but this time acquiring EPOCH_PHOTOMETRY rather than XP_SAMPLED data, and performing some additional processing of the main and array input tables. Only those input rows are used for which the boolean `has_epoch_photometry` column is True; for the linked array tables the rows are limited to those with the `BAND` column having the value "`G`"; and only the array columns `time`, `mag` and `flux` are retained for inclusion into the output.

```
stilts arrayjoin in=obscore-lotssdr2.vot
                 atable='equals(dataproduct_type,"spectrum")?access_url:""'
                 out=with-spectra.vot
```

Takes an ivoa.ObsCore-like table and adds array-valued columns giving the content of spectrum files linked from the `access_url` field. For rows in the input table whose `dataproduct_type` field does not have the value "spectrum" a blank location is substituted insteead of the access_url. That has the effect of ensuring that no attempt is made to add array values for non-spectrum rows of the input table, while retaining those rows in the output. Note all the access_urls in the selected rows must point to spectrum files of a of a similar form, otherwise the command will fail.

B.2 `calc`: Evaluates expressions

`calc` is a very simple utility for evaluating expressions. It uses the same expression evaluator as is used in `tpipe` and the other generic table tasks for things like creating new columns, so it can be used as a quick test to see what expressions work, or in order to evaluate expressions using the various algebraic functions documented in Section 10.6. Since usually no table is involved, you can't refer to column names in the expressions. It has one mandatory parameter, the expression to evaluate, and writes the result to the screen.

B.2.1 Usage

The usage of `calc` is

```
stilts <stilts-flags> calc table=<table>
                           [expression=]<expr>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.Calc`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

expression = <expr> ***(String)***
An expression to evaluate. The functions in Section 10.6 can be used.

table = <table> ***(StarTable)***
A table which provides the context within which `expression` is evaluated. This parameter is optional, and will usually not be required; its only purpose is to allow use of constant expressions (table parameters) associated with the table. These can be referenced using identifiers of the form `param$*`, `ucd$*` or `utype$*` - see Section 10.2 for more detail.

B.2.2 Examples

Here are some examples of using `calc`:

```
stilts calc 1+2
```
Calculates one plus two. Writes "3" to standard output.

```
stilts calc 'isoToMjd("2005-12-25T00:00:00")'
```
Works out the Modified Julian Day corresponding to Christmas 2005. The output is "53729.0".

```
stilts calc 'param$author' table=catalogue.xml
```
In this case the expression is evaluated in the context of the supplied table, which means that the table's parameters can be referenced in the expression. This example just outputs the value of the table parameter named "author".

B.3 `cdsskymatch`: Crossmatches table on sky position against VizieR/SIMBAD table

`cdsskymatch` uses the CDS X-Match service to join a local table to one of the tables hosted by the Centre de Données astronomiques de Strasbourg. This includes all of the VizieR tables and the SIMBAD database. The service is very fast, and in most cases it is the best way to match a local table against a large external table hosted by a service. It is almost certainly much better than using `coneskymatch`, though it is less flexible than TAP (see the `tapquery` task for flexible access to TAP services, and `tapskymatch` for positional matches).

The local table is uploaded to the X-Match service in chunks, and the matches for each chunk are retrieved in turn and eventually stitched together to form the final result. The tool only uploads sky position and an identifier for each row of the input table, but all columns of the input table are reinstated in the result for reference.

For a better understanding of the details of how this service operates, including exactly what coordinates are matched against the uploaded positions (roughly: integrated to J2000 using proper motions if available) and what columns are included in the output (roughly: a subset of the most commonly used columns), please consult the service documentation.

Acknowledgement: CDS note that if the use of the X-Match service is useful to your research, they would appreciate the following acknowledgement:

> *"This research made use of the cross-match service provided by CDS, Strasbourg."*

B.3.1 Usage

The usage of `cdsskymatch` is

```
stilts <stilts-flags> cdsskymatch ifmt=<in-format> istream=true|false
                                  icmd=<cmds> ocmd=<cmds>
                                  omode=out|meta|stats|count|checksum|cgi|discard|topcat|sa
                                  out=<out-table> ofmt=<out-format>
                                  ra=<expr> dec=<expr>
                                  radius=<value/arcsec> cdstable=<value>
                                  find=all|best|best-remote|each|each-dist
                                  blocksize=<int-value> maxrec=<int-value>
                                  compress=true|false
                                  serviceurl=<url-value> usemoc=true|false
                                  presort=true|false fixcols=none|dups|all
                                  suffixin=<label> suffixremote=<label>
                                  [in=]<table>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.CdsUploadSkyMatch`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`blocksize = <int-value>` ***(Integer)***
The CDS Xmatch service operates limits on the maximum number of rows that can be uploaded and the maximum number of rows that is returned as a result from a single query. In the case of large input tables, they are broken down into smaller blocks, and one request is sent to the external service for each block. This parameter controls the number of rows in each block. For an input table with fewer rows than this value, the whole thing is done as a single request.

At time of writing, the maximum upload size is 100Mb (about 3Mrow; this does not depend on the width of your table), and the maximum return size is 2Mrow.

Large blocksizes tend to be good (up to a point) for reducing the total amount of time a large xmatch operation takes, but they can make it harder to see the job progressing. There is also the danger (for ALL-type find modes) of exceeding the return size limit, which will result in truncation of the returned result.

[Default: `50000`]

`cdstable = <value>` ***(String)***
Identifier of the table from the CDS crossmatch service that is to be matched against the local table. This identifier may be the standard VizieR identifier (e.g. "`II/246/out`" for the 2MASS Point Source Catalogue) or "`simbad`" to indicate SIMBAD data.

See for instance the TAPVizieR table searching facility at `http://tapvizier.u-strasbg.fr/adql/` to find VizieR catalogue identifiers.

`compress = true|false` ***(Boolean)***
If true, the service is requested to provide HTTP-level compression for the response stream (Accept-Encoding header is set to "`gzip`", see RFC 2616). This does not guarantee that compression will happen but if the service honours this request it may result in a smaller amount of network traffic at the expense of more processing on the server and client.

[Default: `true`]

`dec = <expr>` ***(String)***
Declination in degrees in the ICRS coordinate system for the position of each row of the input table. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

`find = all|best|best-remote|each|each-dist` ***(UserFindMode)***

Determines which pair matches are included in the result.

- `all`: All matches
- `best`: Matched rows, best remote row for each input row
- `best-remote`: Matched rows, best input row for each remote row
- `each`: One row per input row, contains best remote match or blank
- `each-dist`: One row per input row, column giving distance only for best match

Note only the `all` mode is symmetric between the two tables.

Note also that there is a bug in `best-remote` matching. If the match is done in multiple blocks, it's possible for a remote table row to appear matched against one local table row per uploaded block, rather than just once for the whole result. If you're worried about that, set `blocksize >=` *rowCount*. This may be fixed in a future release.

[Default: `all`]

`fixcols = none|dups|all` ***(Fixer)***

Determines how input columns are renamed before use in the output table. The choices are:

- `none`: columns are not renamed
- `dups`: columns which would otherwise have duplicate names in the output will be renamed to indicate which table they came from
- `all`: all columns will be renamed to indicate which table they came from

If columns are renamed, the new ones are determined by `suffix*` parameters.

[Default: `dups`]

`icmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its

standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istream = true|false` ***(Boolean)***

If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`maxrec = <int-value>` ***(Integer)***

Limit to the number of rows resulting from this operation. If the value is negative (the default) no limit is imposed. Note however that there can be truncation of the result if the number of records returned from a single chunk exceeds the service hard limit (2,000,000 at time of writing).

[Default: `-1`]

`ocmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`

- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`
- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

`out = <out-table>` ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `-`]

`presort = true|false` ***(Boolean)***

If true, the rows are sorted by HEALPix index before they are uploaded to the CDS X-Match service. If the match is done in multiple blocks, this may improve efficiency, since when matching against a large remote catalogue the X-Match service likes to process requests in which sources are grouped into a small region rather than scattered all over the sky.

Note this will have a couple of other side effects that may be undesirable: it will read all the input rows into the task at once, which may make it harder to assess progress, and it will affect the order of the rows in the output table.

It is *probably* only worth setting true for rather large (multi-million-row?) multi-block matches, where both local and remote catalogues are spread over a significant fraction of the sky. But feel free to experiment.

[Default: `false`]

`ra = <expr>` ***(String)***

Right ascension in degrees in the ICRS coordinate system for the position of each row of the input table. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

`radius = <value/arcsec>` ***(Double)***

Maximum distance from the local table (ra,dec) position at which counterparts from the remote table will be identified. This is a fixed value given in arcseconds, and must be in the range [0,180] (this limit is currently enforced by the CDS Xmatch service).

`serviceurl = <url-value>` ***(URL)***

The URL at which the CDS Xmatch service can be found. Normally this should not be altered from the default, but if other implementations of the same service are known, this parameter can be used to access them.

[Default: `http://cdsxmatch.u-strasbg.fr/xmatch/api/v1/sync`]

`suffixin = <label>` ***(String)***

If the `fixcols` parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from the input table.

[Default: `_in`]

`suffixremote = <label>` ***(String)***

If the `fixcols` parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from the CDS result table.

[Default: `_cds`]

`usemoc = true|false` ***(Boolean)***

If true, first acquire a MOC coverage map from CDS, and use that to pre-filter rows before uploading them for matching. This should improve efficiency, but have no effect on the result.

[Default: `true`]

B.3.2 Examples

Here are some examples of `cdsskymatch`:

```
stilts cdsskymatch cdstable=II/246/out find=all
                   in=dr5qso.fits ra=RA dec=DEC radius=1 out=qso_2mass.fits
```

Matches a local catalogue `dr5qso.fits` against the VizieR table `II/246/out` (the 2MASS Point Source Catalogue). The search radius is 1 arcsecond, and all 2MASS sources within the radius of each input source are returned.

```
stilts cdsskymatch cdstable=simbad find=best
                   in=sources.txt ifmt=ascii ra=RAJ2000 dec=DEJ2000 radius=8.5
                   blocksize=1000 icmd=progress omode=topcat
```

This finds the closest object in the SIMBAD database within 8.5 arcsec for each row of an input ASCII table. Uploads are done in blocks of 1,000 rows at a time, and progress is displayed on the console. When the match is complete, the result is sent directly to a running instance of TOPCAT.

```
stilts cdsskymatch in=3XMM_DR4cat_slim_v1.0.fits
                   icmd='select "SC_POSERR < 1 && SC_EXTENT == 0"'
                   cdstable=B/mk/mktypes
                   ra=SC_RA dec=SC_DEC radius=1.5
                   find=best suffixin=_XMM suffixremote=_MK fixcols=all
                   ocmd='select startsWith(spType_MK,\"G\")'
                   out=xmm_gtype.fits
```

This locates XMM-Newton point-like sources identified as being of spectral type G. It uses the 3XMM-DR4 XMM-Newton serendipitous source catalogue as input. The `icmd` filter selects the objects in that catalogue with well-defined point-like positions. It then matches them with Skiff's MK spectral classification catalogue (B/mk/mktypes in VizieR) and finally filters the result to include only those sources identified as being of spectral type G. Thanks to Ada Nebot (CDS) for this example.

B.4 `cone`: Executes a Cone Search-like query

`cone` is a utility to execute one of the "Simple" positional DAL query operations on a remote server: Simple Cone Search, Simple Image Access (SIA) or Simple Spectral Access (SSA). The job it does is not very complicated: given a base URL for a service of one of these types and values for the central position and radius required, it assembles the query URL in the form required for the relevant protocol, retrieves the result of the query, and turns it into a table which can be operated on with the usual STILTS pipeline operations.

B.4.1 Usage

The usage of cone is

```
stilts <stilts-flags> cone serviceurl=<url-value> lon=<degrees>
                          lat=<degrees> radius=<degrees>
                          skysys=icrs|fk5|fk4|galactic|supergalactic|ecliptic
                          servicetype=cone|ssa|sia1|sia2|sia verb=1|2|3
                          compress=true|false dataformat=<value>
                          ocmd=<cmds>
                          omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|plas
                          out=<out-table> ofmt=<out-format>
```

If you don't have the stilts script installed, write "java -jar stilts.jar" instead of "stilts" - see Section 3. The available <stilts-flags> are listed in Section 2.1. For programmatic invocation, the Task class for this command is uk.ac.starlink.ttools.task.TableCone.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

compress = true|false ***(Boolean)***

If true, the service is requested to provide HTTP-level compression for the response stream (Accept-Encoding header is set to "gzip", see RFC 2616). This does not guarantee that compression will happen but if the service honours this request it may result in a smaller amount of network traffic at the expense of more processing on the server and client.

[Default: true]

dataformat = <value> ***(String)***

Indicates the format of data objects described in the returned table. The meaning of this is dependent on the value of the servicetype parameter:

- servicetype=cone: not used
- servicetype=ssa: gives the MIME type of spectra referenced in the output table, also special values "votable", "fits", "compliant", "graphic", "all", and others (value of the SSA FORMAT parameter).
- servicetype=sia1: gives the MIME type required for images/resources referenced in the output table, corresponding to the SIA FORMAT parameter. The special values "GRAPHIC" (all graphics formats) and "ALL" (no restriction) as defined by SIAv1 are also permissible. For SIA version 1 only, this defaults to"image/fits".
- servicetype=sia2: gives the MIME type required for images/resources referenced in the output table, corresponding to the SIA FORMAT parameter. The special values "GRAPHIC" (all graphics formats) and "ALL" (no restriction) as defined by SIAv1 are also permissible.
- servicetype=sia: gives the MIME type required for images/resources referenced in the output table, corresponding to the SIA FORMAT parameter. The special values "GRAPHIC" (all graphics formats) and "ALL" (no restriction) as defined by SIAv1 are also permissible. For SIA version 1 only, this defaults to"image/fits".

lat = <degrees> ***(Double)***

Central latitude position for cone search. By default this is the Declination, but depending on the value of the skysys parameter it may be in a different sky system.

lon = <degrees> ***(Double)***

Central longitude position for cone search. By default this is the Right Ascension, but depending on the value of the skysys parameter it may be in a different sky system.

ocmd = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of

processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`
- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

`out = <out-table>` ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "`-`" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `-`]

`radius = <degrees>` ***(Double)***

Search radius in degrees.

`servicetype = cone|ssa|sia1|sia2|sia` ***(ServiceType)***

Selects the type of data access service to contact. Most commonly this will be the Cone Search service itself, but there are one or two other possibilities:

- `cone`: Cone Search protocol - returns a table of objects found near each location. See

Cone Search standard.
- `ssa`: Simple Spectral Access protocol - returns a table of spectra near each location. See SSA standard.
- `sia1`: Simple Image Access protocol version 1 - returns a table of images near each location. See SIA 1.0 standard.
- `sia2`: Simple Image Access protocol version 2 - returns a table of images near each location. See SIA 2.0 standard.
- `sia`: alias for `sia1`

[Default: `cone`]

`serviceurl = <url-value>` ***(URL)***
The base part of a URL which defines the query to be made. Additional parameters will be appended to this using CGI syntax ("`name=value`", separated by '&' characters). If this value does not end in either a '?' or a '&', one will be added as appropriate.

`skysys = icrs|fk5|fk4|galactic|supergalactic|ecliptic` ***(SkySystem)***
Sky coordinate system used to interpret the `lon` and `lat` parameters. If the value is ICRS (the default) the provided values are assumed to be Right Ascension and Declination and are sent unchanged; for other values they will be converted from the named system into RA and Dec first.

[Default: `icrs`]

`verb = 1|2|3` ***(String)***
Verbosity level of the tables returned by the query service. A value of 1 indicates the bare minimum and 3 indicates all available information.

B.4.2 Examples

Here are some examples of `cone`:

```
stilts cone serviceurl=http://gaia.ari.uni-heidelberg.de/cone/search
           lon=56.75 lat=24.12 radius=0.8
           out=pleiades.fits
```

Queries the ARI-Gaia cone search service for sources within 0.8 of the given sky position, and writes the result to a file.

```
stilts cone serviceurl=http://gaia.ari.uni-heidelberg.de/cone/search
           lon=56.75 lat=24.12 radius=0.8
           verb=1
           ocmd='sorthead 10 phot_g_mean_mag'
           ocmd='keepcols "source_id ra dec phot_g_mean_mag"'
```

This does the same basic query as the previous example, but post-processes the result so that a limited amount of data (source identifier, position and magnitude) for only the ten brightest sources is written to the console. Since most of the columns are discarded, we specify `verb=1` which indicates to the service that only a minimal column set is required in the query result.

```
stilts -verbose
       cone servicetype=ssa serviceurl='http://archive.eso.org/ssap'
            lon=0 lat=90 radius=1.0 skysys=galactic
            omode=count
```

Queries the ESO Simple Spectral Access service for spectra within one degree of the northern galactic pole. The `omode=count` parameter means that it just counts the rows and columns and prints the numbers to the console. The `-verbose` flag means that (amongst other things) the full URL that the command used to make the query will be logged to the console.

B.5 `coneskymatch`: Crossmatches table on sky position against remote cone service

Note: this command is very inefficient for large tables, and in most cases `cdsskymatch` or `tapskymatch` provide better alternatives.

`coneskymatch` is a utility which performs a cone search-like query to a remote server for each row of an input table. Each of these queries returns a table with one row for each item held by the server in the region of sky represented by the input row. The results of all the queries are then concatenated into one big output table which is the output of this command.

The type of virtual observatory service queried is determined by the `servicetype` parameter. Typically it will be a Cone Search service, which queries a remote catalogue for astronomical objects or sources in a particular region. However, you can also query Simple Image Access and Simple Spectral Access services in just the same way, to return tables of available image and spectral resources in the relevant regions.

The identity of the server to query is given by the `serviceurl` parameter. Some advice about how to locate URLs for suitable services is given in Appendix B.5.3.

The effect of this command is like doing a positional crossmatch where one of the catalogues is local and the other is remote and exposes its data via a cone search/SIA/SSA service. Because of both the network communication and the necessarily naive crossmatching algorithm (which scales linearly with the size of the local catalogue) however, it is only suitable if the local catalogue has a reasonably small number of rows, unless you are prepared to wait a long time.

The `parallel` parameter allows you to perform multiple cone searches concurrently, so that instead of completing the first cone search, then the second, then the third, the program can be executing a number of them at once. This can speed up operation considerably, especially in the face of network latency, but beware that submitting a very large number of queries simultaneously to the same server may overload it, resulting in some combination of failed queries, ultimately slower runtimes, and unpopularity with server admins. Best to start with a low parallelism and cautiously increase it to see whether there are gains in performance.

Note that when running, `coneskymatch` can generate a lot of WARNING messages. Most of these are complaining about badly formed VOTables being returned from the cone search services. STILTS does its best to work out what the service responses mean in this case, and usually makes a good enough job of it.

Note: this task was known as `multicone` in its experimental form in STILTS v1.2 and v1.3.

B.5.1 Usage

The usage of `coneskymatch` is

```
stilts <stilts-flags> coneskymatch ifmt=<in-format> istream=true|false
                                   icmd=<cmds> ocmd=<cmds>
                                   omode=out|meta|stats|count|checksum|cgi|discard|topcat|s
                                   out=<out-table> ofmt=<out-format>
                                   ra=<expr> dec=<expr> sr=<expr/deg>
                                   find=best|all|each usefoot=true|false
                                   footnside=<int-value>
                                   copycols=<colid-list>
                                   scorecol=<col-name> parallel=<n>
                                   erract=abort|ignore|retry|retry<n>
                                   ostream=true|false fixcols=none|dups|all
                                   suffix0=<label> suffix1=<label>
                                   servicetype=cone|ssa|sia1|sia2|sia
```

```
serviceurl=<url-value> verb=1|2|3
dataformat=<value> emptyok=true|false
compress=true|false
[in=]<table>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.MultiCone`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`compress = true|false` ***(Boolean)***
If true, the service is requested to provide HTTP-level compression for the response stream (Accept-Encoding header is set to "`gzip`", see RFC 2616). This does not guarantee that compression will happen but if the service honours this request it may result in a smaller amount of network traffic at the expense of more processing on the server and client.

[Default: `true`]

`copycols = <colid-list>` ***(String)***
List of columns from the input table which are to be copied to the output table. Each column identified here will be prepended to the columns of the combined output table, and its value for each row taken from the input table row which provided the parameters of the query which produced it. See Section 6.3 for list syntax. The default setting is "`*`", which means that all columns from the input table are included in the output.

[Default: `*`]

`dataformat = <value>` ***(String)***
Indicates the format of data objects described in the returned table. The meaning of this is dependent on the value of the `servicetype` parameter:

- `servicetype=cone`: not used
- `servicetype=ssa`: gives the MIME type of spectra referenced in the output table, also special values "`votable`", "`fits`", "`compliant`", "`graphic`", "`all`", and others (value of the SSA FORMAT parameter).
- `servicetype=sia1`: gives the MIME type required for images/resources referenced in the output table, corresponding to the SIA FORMAT parameter. The special values "`GRAPHIC`" (all graphics formats) and "`ALL`" (no restriction) as defined by SIAv1 are also permissible. For SIA version 1 only, this defaults to "`image/fits`".
- `servicetype=sia2`: gives the MIME type required for images/resources referenced in the output table, corresponding to the SIA FORMAT parameter. The special values "`GRAPHIC`" (all graphics formats) and "`ALL`" (no restriction) as defined by SIAv1 are also permissible.
- `servicetype=sia`: gives the MIME type required for images/resources referenced in the output table, corresponding to the SIA FORMAT parameter. The special values "`GRAPHIC`" (all graphics formats) and "`ALL`" (no restriction) as defined by SIAv1 are also permissible. For SIA version 1 only, this defaults to "`image/fits`".

`dec = <expr>` ***(String)***
Declination in degrees in the ICRS coordinate system for the position of each row of the input table. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

`emptyok = true|false` ***(Boolean)***
Whether the table metadata which is returned from a search result with zero rows is to be believed. According to the spirit, though not the letter, of the cone search standard, a cone

search service which returns no data ought nevertheless to return the correct column headings. Unfortunately this is not always the case. If this parameter is set `true`, it is assumed that the service behaves properly in this respect; if it does not an error may result. In that case, set this parameter `false`. A consequence of setting it false is that in the event of no results being returned, the task will return no table at all, rather than an empty one.

[Default: `true`]

`erract = abort|ignore|retry|retry<n>` ***(ConeErrorPolicy)***

Determines what will happen if any of the individual cone search requests fails. By default the task aborts. That may be the best thing to do, but for unreliable or poorly implemented services you may find that some searches fail and others succeed so it can be best to continue operation in the face of a few failures. The options are:

- `abort`: Failure of any query terminates the task.
- `ignore`: Failure of a query is treated the same as a query which returns no rows.
- `retry`: Failed queries are retried until they succeed; an increasing delay is introduced for each failure. Use with care - if the failure is for some good, or at least reproducible reason this could prevent the task from ever completing.
- `retry<n>`: Failed queries are retried at most a fixed number `<n>` of times; an increasing delay is introduced for each failure. If failures persist the task terminates.

[Default: `abort`]

`find = best|all|each` ***(String)***

Determines which matches are retained.

- `best`: Only the matching query table row closest to the input table row will be output. Input table rows with no matches will be omitted. (Note this corresponds to the `best1` option in the pair matching commands, and `best1` is a permitted alias).
- `all`: All query table rows which match the input table row will be output. Input table rows with no matches will be omitted.
- `each`: There will be one output table row for each input table row. If matches are found, the closest one from the query table will be output, and in the case of no matches, the query table columns will be blank.

[Default: `all`]

`fixcols = none|dups|all` ***(Fixer)***

Determines how input columns are renamed before use in the output table. The choices are:

- `none`: columns are not renamed
- `dups`: columns which would otherwise have duplicate names in the output will be renamed to indicate which table they came from
- `all`: all columns will be renamed to indicate which table they came from

If columns are renamed, the new ones are determined by `suffix*` parameters.

[Default: `dups`]

`footnside = <int-value>` ***(Integer)***

Determines the HEALPix Nside parameter for use with the MOC footprint service. This tuning parameter determines the resolution of the footprint if available. Larger values give better resolution, hence a better chance of avoiding unnecessary queries, but processing them takes longer and retrieving and storing them is more expensive.

The value must be a power of 2, and at the time of writing, the MOC service will not supply footprints at resolutions greater than nside=512, so it should be <=512.

Only used if `usefoot=true`.

`icmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istream = true|false` ***(Boolean)***

If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`ocmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ofmt = <out-format> ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "(auto)" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if omode has its default value of "out".

[Default: (auto)]

omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is out, which means that the result will be written as a new table to disk or elsewhere, as determined by the out and ofmt parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (<mode-args>) are required to determine the exact behaviour.

Possible values are

- out
- meta
- stats
- count
- checksum
- cgi
- discard
- topcat
- samp
- plastic
- tosql
- gui

Use the help=omode flag or see Section 6.4 for more information.

[Default: out]

ostream = true|false ***(Boolean)***

If set true, this will cause the operation to stream on output, so that the output table is built up as the results are obtained from the cone search service. The disadvantage of this is that some output modes and formats need multiple passes through the data to work, so depending on the output destination, the operation may fail if this is set. Use with care (or be prepared for the operation to fail).

[Default: false]

out = <out-table> ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if omode has its default value of "out".

[Default: -]

parallel = <n> ***(Integer)***

Allows multiple cone searches to be performed concurrently. If set to the default value, 1, the cone query corresponding to the first row of the input table will be dispatched, when that is completed the query corresponding to the second row will be dispatched, and so on. If set to <n>, then queries will be overlapped in such a way that up to approximately <n> may be running at any one time.

Whether increasing <n> is a good idea, and what might be a sensible maximum value, depends on the characteristics of the service being queried. In particular, setting it to too large a number may overload the service resulting in some combination of failed queries, ultimately slower runtimes, and unpopularity with server admins.

The maximum value permitted for this parameter by default is 5. This limit may be raised by use of the service.maxparallel system property but use that option with great care since you may overload services and make yourself unpopular with data centre admins. As a rule, you should only increase this value if you have obtained permission from the data centres whose services on which you will be using the increased parallelism.

[Default: 1]

ra = <expr> ***(String)***

Right ascension in degrees in the ICRS coordinate system for the position of each row of the input table. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

scorecol = <col-name> ***(String)***

Gives the name of a column in the output table to contain the distance between the requested central position and the actual position of the returned row. The distance returned is an angular distance in degrees. If a null value is chosen, no distance column will appear in the output table.

[Default: Separation]

servicetype = cone|ssa|sia1|sia2|sia ***(ServiceType)***

Selects the type of data access service to contact. Most commonly this will be the Cone Search service itself, but there are one or two other possibilities:

- cone: Cone Search protocol - returns a table of objects found near each location. See Cone Search standard.
- ssa: Simple Spectral Access protocol - returns a table of spectra near each location. See SSA standard.
- sia1: Simple Image Access protocol version 1 - returns a table of images near each location. See SIA 1.0 standard.
- sia2: Simple Image Access protocol version 2 - returns a table of images near each location. See SIA 2.0 standard.
- sia: alias for sia1

[Default: cone]

serviceurl = <url-value> ***(URL)***

The base part of a URL which defines the queries to be made. Additional parameters will be appended to this using CGI syntax ("name=value", separated by '&' characters). If this value does not end in either a '?' or a '&', one will be added as appropriate.

See Appendix B.5.3 for discussion of how to locate service URLs corresponding to given datasets.

sr = <expr/deg> ***(String)***

Expression which evaluates to the search radius in degrees for the request at each row of the input table. This will often be a constant numerical value, but may be the name or ID of a column in the input table, or a function involving one.

suffix0 = <label> ***(String)***

If the fixcols parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from the input table.

[Default: _0]

suffix1 = <label> ***(String)***

If the fixcols parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from the cone result table.

[Default: _1]

usefoot = true|false ***(Boolean)***

Determines whether an attempt will be made to restrict searches in accordance with available footprint information. If this is set true, then before any of the per-row queries are performed, an attempt may be made to acquire footprint information about the servce. If such information can be obtained, then queries which fall outside the footprint, and hence which are known to yield no results, are skipped. This can speed up the search considerably.

Currently, the only footprints available are those provided by the CDS MOC (Multi-Order Coverage map) service, which covers VizieR and a few other cone search services.

[Default: true]

verb = 1|2|3 ***(String)***

Verbosity level of the tables returned by the query service. A value of 1 indicates the bare minimum and 3 indicates all available information.

B.5.2 Examples

Here are some examples of coneskymatch:

```
stilts coneskymatch serviceurl=http://archive.stsci.edu/hst/search.php \
                    in=messier.xml sr=0.05 out=matches.xml
```

This queries the HST cone search service from Space Telescope for records within .05 degrees of each Messier object contained in a local VOTable messier.xml. The sky positions in the input catalogue are guessed from the available table metadata. The result is written to a new VOTable, matches.xml. Since the servicetype parameter is not given, the default (cone search) service type is assumed.

```
stilts coneskymatch
       servicetype=sia \
       serviceurl=http://irsa.ipac.caltech.edu/cgi-bin/2MASS/IM/nph-im_sia?type=ql&ds=asky \
       in=messier.xml ra=RA dec=DEC \
       dataformat=image/fits \
       out=fitsimages.xml
```

This is similar to the previous example, but instead of querying an HST cone search server for catalogue objects near the input table positions, it queries a 2MASS Simple Image Access (SIA) server for images. It also explicitly names the columns holding the J2000 positions of reach record in the input catalogue as RA and DEC. The search radius parameter (sr) is not set here; for SIA queries the default search radius is zero, which has the special meaning of including any image which covers the requested position. Setting dataformat=image/fits (which is the default) requests only records describing FITS-format images to be returned; setting it to an empty value might return other formats such as JPEG too.

```
stilts coneskymatch \
       serviceurl='http://www.nofs.navy.mil/cgi-bin/vo_cone.cgi?CAT=NOMAD' \
       in=vizier.xml#7 \
       icmd='addskycoords -inunit sex fk4 fk5 RAB1950 DEB1950 RAJ2000 DEJ2000' \
       icmd='progress'
       ra=RAJ2000 dec=DEJ2000 sr=0.01 \
       ocmd='replacecol -units deg RA hmsToDegrees(RA[0],RA[1],RA[2])' \
       ocmd='replacecol -units deg DEC dmsToDegrees(DEC[0],DEC[1],DEC[2])' \
       omode=topcat
```

In this example some pre-processing of the input catalogue and post-processing of the output catalogue is performed as well as the multiple cone search itself.

The input catalogue, which is the 8th TABLE element in a VOTable file, contains sky positions in sexagesimal FK4 (B1950) coordinates. The `icmd=addskycoords...` parameter specifies a filter which will add new columns in FK5 (J2000) degrees, which are what the `coneskymatch` command requires. The `icmd=progress` parameter specifies a filter which will write progress information to the terminal so you can see how the queries are progressing.

The NOMAD service specified by the `serviceurl` parameter used here happens to return results with the RA/DEC columns represented in a rather eccentric format, namely 3-element floating point arrays representing (hours,minutes,seconds)/(degrees,minutes,seconds). The two `ocmd=replacecol...` filters replace the values of these columns with the scalar equivalents in degrees. Finally, the `omode=topcat` parameter causes the result table to be loaded directly into TOPCAT (if it is available).

```
stilts coneskymatch serviceurl='http://archive.stsci.edu/iue/search.php?' \
                    in=queries.txt ifmt=ascii \
                    ra='$1' dec='$2' \
                    sr='$3' copycols='$4' \
                    out=found.fits
```

Here the input is a plain text table with four unnamed columns, giving in order the right ascension, declination, positional error and name of target objects. The command carries out a cone search to the named service for each one. Note in this case the search radius (`sr` parameter) is taken from the table and so varies for each query. The `copycols` parameter has the value '`$4`', which means that the value of the fourth column of the input table will be prepended to each row of the output table for which it is responsible. Output is to a FITS table.

B.5.3 Locating Cone Query Service URLs

To use the `coneskymatch` command you need the **service URL** (also known as the **base URL** or **access URL**) of a cone search, SIA or SSA service to use. If you know one of these representing a service that you wish to use, you can use it directly.

If you don't, you will need to find the URL from somewhere. It is the job of the Virtual Observatory **Registry** to keep a record of where you can find various astronomical services, so this is where you should look.

There are various ways you can interrogate the registry; the easiest is probably to use a graphical registry search tool. One such tool is AstroGrid's **VOExplorer**, which allows you to perform sophisticated searches for cone search, SIA or SSA services. Another option is to use TOPCAT; the Cone Search, SIA and SSA load dialogues allow you to search the registry for these services prior to performing a query; you can just use the registry part and cut'n'paste the URL which is shown.

Other registry querying tools are available, including STILTS's `regquery` (Appendix B.18) command. See that section of the manual for details, but for instance to locate registered Cone Search services which have something to do with SDSS data, you could execute the following:

```
stilts regquery query="capability/@standardID = 'ivo://ivoa.net/std/ConeSearch' and title
       ocmd="keepcols 'shortName AccessUrl'" \
       ofmt=ascii
```

Writing just `query="capability/@standardID = 'ivo://ivoa.net/std/ConeSearch'"` with no further qualification would give you *all* registered cone search services.

B.6 `datalinklint`: Validates DataLink documents

`datalinklint` runs a series of tests on a VOTable that is supposed to conform to the *{links}-response* format defined in the IVOA DataLink specification, and reports the results. This is not likely to be of use to normal users, it is intended for people developing or operating DataLink-compliant services to assess correctness, perhaps with a view to improving compliance.

To run it, you point it at the URL or filename of a VOTable of interest, and it runs various tests on it and reports them to standard output. Only the input table itself, and if the input location uses HTTP the HTTP headers, are tested, this validator does not inspect any other resources. As well as validation checks, some reporting of the table's content (such as a summary of the link defined by each row and a listing of the defined service descriptors) is provided as INFO-level output for convenience. This can be suppressed if preferred by use of the `report` parameter.

This operates in much the same way as the `taplint` command, and the output has a similar format, though unlike `taplint` this command does not divide the testing up into stages. Each report line is of the form:

```
T-MMMMxN aaaaa...
```

where the parts have the following meanings:

- `T`: Report type, one of E(rror), W(arning), I(nfo), S(ummary), F(ailure). See the documentation of the `report` parameter for further description of what these mean. The `report` parameter can be used to suppress some of these; only `E` indicates actual service compliance errors, but including the others may make it easier to see what's going on.
- `MMMM`: Message label, which is always the same for messages generated by the same test, is usually different for messages generated by different tests, and may be somewhat mnemonic.
- `x`: Continuation indicator, either "`-`" or "`+`". In most cases it is "`-`", indicating the first line of a message, but multi-line messages (rare) use "`-`" for the first line and "`+`" for any continuation lines.
- `N`: Sequence number, which is 1 for the first time message `T-MMMM` is reported, and increases by one for each subsequent appearance. After a certain maximum (determined by the `maxrepeat` parameter) additional reports with the same code are no longer output individually, but a summary of the number of reports so discarded is written at the end of the section with the character "`x`" instead of the sequence number. This behaviour prevents the output being swamped by multiple reports of the same issue. If the `maxrepeat` parameter is increased above 9, more than one digit will be used here (so e.g. for maxrepeat=999, the format would be `NNN` not `N`).
- `aaaaa...`: Message text, a free text description of what is being reported.

If you don't like that format, others may be selected using the `format` parameter, which currently also supports JSON. For more flexible interaction with the output you can invoke `taplint` programmatically and supply your own `OutputReporter` instance.

As with any validator, this command does not guarantee to pick up everything wrong with the indicated VOTable, but it tries as hard as it can to check against the rules set out in the DataLink specification and other related documents as appropriate. A non-exhaustive list of the items checked is:

- VOTable validation *à la* `votlint`
- Correct RESOURCE structure of VOTable document
- All required columns are present with mandated metadata
- Each row contains one of `access_url`, `service_def`, `error_message`
- Service descriptors correctly specified and referenced
- Syntax of `error_message` column values
- Syntax of `content_type` column values
- Syntax of `semantics` column values

- Reported HTTP content-type, if applicable
- Correct serialization (TABLEDATA) is used
- DALI error-response checking, if applicable

At present **no checking** is performed on the link targets; tests are confined to the presented document itself.

HTTP connections made by this validator are flagged in the `User-Agent` field with the token "`(IVOA-test)`".

B.6.1 Usage

The usage of `datalinklint` is

```
stilts <stilts-flags> datalinklint format=text|json report=[EWISF]+
                                   maxrepeat=<int-value>
                                   truncate=<int-value> debug=true|false
                                   [votable=]<filename>|<URL>|-
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.DatalinkLint`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`debug = true|false` ***(Boolean)***
If true, debugging output including stack traces will be output along with the normal validation messages.

[Default: `false`]

`format = text|json` ***(OutputReporter)***
Determines the format of the output. Possible values are `text`, `json`.

Note not all of the other parameters may be applicable to all output formats.

[Default: `text`]

`maxrepeat = <int-value>` ***(Integer)***
Puts a limit on the number of times that a single message will be repeated. By setting this to some reasonably small number, you can ensure that the output does not get cluttered up by millions of repetitions of essentially the same error.

[Default: `9`]

`report = [EWISF]+` ***(String)***
Letters indicating which message types should be listed. Each character of the string is one of the letters `E`, `W`, `I`, `S`, `F` with the following meanings:

- `E`: Error in operation or standard compliance of the service.
- `W`: Warning that service behaviour is questionable, or contravenes a standard recommendation, but is not in actual violation of the standard.
- `I`: Information about progress, for instance details of queries made.
- `S`: Summary of previous successful/unsuccessful reports.
- `F`: Failure of the validator to perform some testing. The cause is either some error internal to the validator, or some error or missing functionality in the service which has already been reported.

[Default: `EWISF`]

`truncate = <int-value>` ***(Integer)***

Limits the line length written to the output.

[Default: `640`]

`votable = <filename>|<URL>|-` ***(String)***

Location of the DataLink VOTable document to check. This may be a URL, or a filename, or the special value "`-`" to indicate standard input.

B.6.2 Examples

Here are some examples of `datalinklint`:

`stilts datalinklint votable=links-response.vot`

Perfoms validation on a VOTable file that has been downloaded from a DataLink links service, and reports the results to standard output.

`stilts datalinklint votable=http://example.org/links?ID=2112`
`truncate=80 report=EW`

Validates a links response directly from the server, testing the HTTP transport requirements as well as the content. Output lines are truncated to 80 characters for convenience, and only the E(rror) and W(arning) lines are displayed.

The output of this invocation might look like:

```
This is the STILTS DataLink validator, 3.1-3
E-DLCT-1 Incorrect Content-Type text/xml for DataLink service http://example....
E-RUCD-1 Wrong UCD for column service_def; meta.code != meta.ref
E-SMCO-1 Unknown predicate 'alternative' from core DataLink vocabulary at row 1
E-SMCO-2 Unknown predicate 'alternative' from core DataLink vocabulary at row 2

Totals: Errors: 4; Warnings: 0
```

To output in JSON format instead, you could specify `format=json`.

B.7 `funcs`: Browses functions used by algebraic expression language

`funcs` is a utility which allows you to browse the functions you can use in STILTS's algebraic expression language. Invoking the command causes a window to pop up on the display with two parts. The left hand panel contains a tree-like representation of the functions available - the top level shows the classes (categories) into which the functions are divided, and if you open these up (by double clicking on them) each contains a list of functions and constants in that class. If you click on any of these classes or their constituent functions or constants, a full descritption of what they are and how to use them will appear in the right hand panel.

The information available from this command is the same as that given in Section 10.6, but the graphical browser may be a more convenient way to view the documentation. There are no parameters.

B.7.1 Usage

The usage of `funcs` is

```
stilts <stilts-flags> funcs
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" -

see Section 3. The available <stilts-flags> are listed in Section 2.1. For programmatic invocation, the Task class for this command is uk.ac.starlink.ttools.task.ShowFunctions.

This task has no parameters.

B.8 pixfoot: Generates Multi-Order Coverage maps

pixfoot takes a list of sky positions from an input table and generates a pixel map describing a sky region which includes them all. Currently the output is to a format known as a Multi-Order Coverage map (MOC), which is a HEALPix-based format composed of a list of HEALPix pixels of different sizes, which can efficiently describe complex regions. Other output formats may be introduced in the future.

See also the Coverage class for MOC-related functions.

B.8.1 Usage

The usage of pixfoot is

```
stilts <stilts-flags> pixfoot ifmt=<in-format> istream=true|false
                              icmd=<cmds> order=<int-value> ra=<expr>
                              dec=<expr> radius=<expr> mocfmt=fits|json
                              out=<out-file>
                              [in=]<table>
```

If you don't have the stilts script installed, write "java -jar stilts.jar" instead of "stilts" - see Section 3. The available <stilts-flags> are listed in Section 2.1. For programmatic invocation, the Task class for this command is uk.ac.starlink.ttools.task.PixFootprint.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

dec = <expr> ***(String)***
Declination in degrees for the position of each row of the input table. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

icmd = <cmds> ***(ProcessingStep[])***
Specifies processing to be performed on the input table as specified by parameter in, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ifmt = <in-format> ***(String)***
Specifies the format of the input table as specified by parameter in. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value (auto) (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istream = true|false` ***(Boolean)***
If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`mocfmt = fits|json` ***(MocFormat)***
Determines the output format for the MOC file.

[Default: `fits`]

`order = <int-value>` ***(Integer)***
Maximum HEALPix order for the MOC. This defines the maximum resolution of the output coverage map. The angular resolution corresponding to order *k* is approximately 180/sqrt(3.Pi)/2^*k* (3520*2^-*k* arcmin).

[Default: `13`]

`out = <out-file>` ***(uk.ac.starlink.util.Destination)***
The location of the output file. This is usually a filename to write to. If it is equal to the special value "-" the output will be written to standard output.

[Default: `-`]

`ra = <expr>` ***(String)***
Right ascension in degrees for the position of each row of the input table. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

`radius = <expr>` ***(String)***
Expression which evaluates to the radius in degrees of the cone at each row of the input table. The default is "`0`", which treats each position as a point rather than a cone, but a constant or an expression as described in Section 10 may be used instead.

[Default: `0`]

B.8.2 Examples

Here are some examples of `pixfoot`:

```
stilts pixfoot in=survey.vot order=8 mocfmt=fits out=sfoot.fits
```

Generates an order-8 FITS MOC file from the point positions of rows in the given VOTable. The columns representing sky position are determined automatically (if possible) by examining the metadata in the input table.

```
stilts pixfoot in='jdbc:mysql://localhost/astro1#SELECT * FROM first1'
               icmd='addskycoords galactic icrs GLON GLAT ALPHA DELTA'
               ra=ALPHA dec=DELTA radius=20./3600.
               order=13 mocfmt=fits out=first.moc
```

Generates an order-13 FITS MOC file from positions in a table held in a database. The positions in the original table are in galactic coordinates, so have to be converted to equatorial (ICRS) first. The map is formed in this case by surrounding each point by a disc of 20 arcsec. Note that JDBC database access will have to be set up as per Section 3.4 for this command to work.

B.9 `pixsample`: Samples from a HEALPix pixel data file

`pixsample` samples data at the sky position represented by each row from an all-sky map contained in a HEALPix-format pixel data file. Such files are actually tables (usually in FITS format) in which the row number corresponds to a HEALPix pixel index, and the pixel values are cell contents; one or more columns may be present containing values for one or more all-sky maps. The result of this command is to add a column to the input table representing the pixel data at the position of each input row for each of the data columns in the HEALPix table.

This command does not attempt to convert between coordinate systems except as instructed, so it is important to know what coordinate system the HEALPix file is in, and ensure that the coordinates supplied from the input table match this. You may need to examine the documentation or headers of the HEALPix file in question to find out. See the Examples section for some examples.

There is a choice of how the sampling is done; the simplest way is just to use the value of the pixel covering the indicated position. An alternative is to average over a disc of given radius (perhaps a function of the input row). Other options (e.g. max/min) could easily be added.

Although HEALPix is not a common format for storing image data in general, it is used for storing a number of important all-sky data sets such as the WMAP results and Schlegel dust maps. The NASA LAMBDA (http://lambda.gsfc.nasa.gov/) (Legacy Archive for Microwave Background Data Analysis) archive has a number of maps in a suitable format, including foreground data like predicted reddening as well as CMB maps.

Note at present this command only supports all-sky, not partial, HEALPix maps. Partial map support may be added at some point in the future if there is demand.

B.9.1 Usage

The usage of `pixsample` is

```
stilts <stilts-flags> pixsample in=<table> ifmt=<in-format> icmd=<cmds>
                               pixdata=<pix-table> pfmt=<in-format>
                               pcmd=<cmds> ocmd=<cmds>
                               omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp
                               out=<out-table> ofmt=<out-format>
                               pixorder=nested|ring|(auto) stat=point|mean
                               lon=<expr> lat=<expr>
                               insys=icrs|fk5|fk4|galactic|supergalactic|ecliptic
                               pixsys=icrs|fk5|fk4|galactic|supergalactic|ecliptic
```

```
radius=<expr>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.PixSample`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`icmd = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`insys = icrs|fk5|fk4|galactic|supergalactic|ecliptic` ***(SkySystem)***
Specifies the sky coordinate system in which sample positions are provided by the `lon`/`lat` parameters. If the sample positions are given in the same coordinate system as that given by the pixel data table, both the `insys` and `pixsys` parameters may be set `null`.

The available coordinate systems are:

- `icrs`: ICRS (Right Ascension, Declination)
- `fk5`: FK5 J2000.0 (Right Ascension, Declination)
- `fk4`: FK4 B1950.0 (Right Ascension, Declination)
- `galactic`: IAU 1958 Galactic (Longitude, Latitude)
- `supergalactic`: de Vaucouleurs Supergalactic (Longitude, Latitude)
- `ecliptic`: Ecliptic (Longitude, Latitude)

`lat = <expr>` ***(String)***

Expression which evaluates to the latitude coordinate in degrees in the input table at which positions are to be sampled from the pixel data table. This will usually be the name or ID of a column in the input table, or an expression involving one. If this coordinate does not match the coordinate system used by the pixel data table, both coordinate systems must be set using the `insys` and `pixsys` parameters.

`lon = <expr>` ***(String)***

Expression which evaluates to the longitude coordinate in degrees in the input table at which positions are to be sampled from the pixel data table. This will usually be the name or ID of a column in the input table, or an expression involving one. If this coordinate does not match the coordinate system used by the pixel data table, both coordinate systems must be set using the `insys` and `pixsys` parameters.

`ocmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`

- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

`out = <out-table>` ***(TableConsumer)***
The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `-`]

`pcmd = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on pixel data table as specified by parameter `pixdata`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`pfmt = <in-format>` ***(String)***
File format for the HEALPix pixel data table. This is usually, but not necessarily, FITS.

[Default: `(auto)`]

`pixdata = <pix-table>` ***(StarTable)***
The location of the table containing the pixel data. The data must be in the form of a HEALPix table, with one pixel per row in HEALPix order. These files are typically, but not necessarily, FITS tables. A filename or URL may be used, but a local file will be more efficient.

Some HEALPix format FITS tables seem to have rows which contain 1024-element arrays of pixels instead of single pixel values. This (rather perverse?) format is not currently supported here, but if there is demand support could be added.

`pixorder = nested|ring|(auto)` ***(HealpixScheme)***
Selects the pixel ordering scheme used by the pixel data file. There are two different ways of ordering pixels in a HEALPix file, "ring" and "nested", and the sampler needs to know which one is in use. If you know which is in use, choose the appropriate value for this parameter; if `(auto)` is used it will attempt to work it out from headers in the file (the ORDERING header). If no reliable ordering scheme can be determined, the command will fail with an error.

[Default: `(auto)`]

`pixsys = icrs|fk5|fk4|galactic|supergalactic|ecliptic` ***(SkySystem)***
Specifies the sky coordinate system used for the HEALPix data in the pixdata file. If the sample positions are given in the same coordinate system as that given by the pixel data table, both the `insys` and `pixsys` parameters may be set `null`.

The available coordinate systems are:

- `icrs`: ICRS (Right Ascension, Declination)
- `fk5`: FK5 J2000.0 (Right Ascension, Declination)
- `fk4`: FK4 B1950.0 (Right Ascension, Declination)
- `galactic`: IAU 1958 Galactic (Longitude, Latitude)
- `supergalactic`: de Vaucouleurs Supergalactic (Longitude, Latitude)
- `ecliptic`: Ecliptic (Longitude, Latitude)

radius = <expr> ***(String)***

Determines the radius in degrees over which pixels will be sampled to generate the output statistic in accordance with the value of the `stat` parameter. This will typically be a constant value, but it may be an algebraic expression based on columns from the input table.

Not used if `stat=point`.

stat = point|mean ***(StatMode)***

Determines how the pixel values will be sampled to generate an output value. The options are:

- `point`: Uses the value at the pixel covering the supplied position. In this case the `radius` parameter is not used.
- `mean`: Averages the values over all the pixels within a radius given by the `radius` parameter. This averaging is somewhat approximate; all pixels which are mostly within the radius are averaged with equal weights.

[Default: `point`]

B.9.2 Examples

Here are some examples of `pixsample`:

```
stilts pixsample in=szdata.fits pixdata=wmap_ilc_7yr_v4.fits
                 lat=GAL_LAT lon=GAL_LON pcmd='keepcols TEMPERATURE'
                 out=szdata_cmb.fits
```

Samples from a HEALPix file containing WMAP data are added to an input file `szdata.fits`, giving an output file `szdata_cmb.fits` which is the same but with an additional column `TEMPERATURE`. The sampling is done using the default statistical mode `point`, which just takes a point sample at the input position. The HEALPix file must have its pixels ordered using galactic coordinates, since that is the coordinate system available from the input table.

The pixdata file used here can be found (at time of writing) at http://lambda.gsfc.nasa.gov/data/map/dr4/dfp/ilc/wmap_ilc_7yr_v4.fits (24 Mbyte).

```
stilts pixsample in=messier.xml pixdata=lambda_sfd_ebv.fits
                 stat=mean radius=5./60.
                 insys=icrs pixsys=galactic lon=RA2000 lat=DEC2000
```

Samples data from a HEALPix table, averaging over a sampling radius of 5 arcmin. The coordinates in the input table are only available as ICRS (RA,Dec) coordinates, and the arrangement of the HEALPix pixels in the pixel data file uses galactic coordinates (you can only determine this by looking at the FITS headers or documentation of that file), so it is necessary to use the `insys` and `pixsys` parameters for conversion.

The pixdata file used here can be found (at time of writing) at http://lambda.gsfc.nasa.gov/data/foregrounds/SFD/lambda_sfd_ebv.fits (25 Mbyte).

B.10 `plot2plane`: Draws a plane plot

`plot2plane` draws plots on a Cartesian 2-dimensional surface.

Positional coordinates are specified as `x`, `y` pairs, e.g.:

```
plot2plane layer1=mark in1=cat.fits x1=RMAG y1=RMAG-BMAG
```

Content is added to the plot by specifying one or more *plot layers* using the `layerN` parameter. The

N part is a suffix applied to all the parameters affecting a given layer; any suffix (including the empty string) may be used. Available layers for this plot type are: mark (Section 8.3.1), size (Section 8.3.2), sizexy (Section 8.3.3), xyvector (Section 8.3.4), xyerror (Section 8.3.5), xyellipse (Section 8.3.6), xycorr (Section 8.3.7), link2 (Section 8.3.8), mark2 (Section 8.3.9), poly4 (Section 8.3.10), mark4 (Section 8.3.11), polygon (Section 8.3.12), area (Section 8.3.13), central (Section 8.3.14), lines (Section 8.3.15), marks (Section 8.3.16), yerrors (Section 8.3.17), xyerrors (Section 8.3.18), statline (Section 8.3.19), statmark (Section 8.3.20), arrayquantile (Section 8.3.21), line (Section 8.3.22), linearfit (Section 8.3.23), label (Section 8.3.24), arealabel (Section 8.3.25), contour (Section 8.3.26), grid (Section 8.3.27), fill (Section 8.3.28), quantile (Section 8.3.29), histogram (Section 8.3.30), kde (Section 8.3.31), knn (Section 8.3.32), densogram (Section 8.3.33), gaussian (Section 8.3.34), function (Section 8.3.35).

B.10.1 Usage

The usage of plot2plane is

```
stilts <stilts-flags> plot2plane xpix=<int-value> ypix=<int-value>
                                 insets=<top>,<left>,<bottom>,<right>
                                 omode=swing|out|cgi|discard|auto
                                 storage=simple|memory|disk|policy|cache|basic-cache|persis
                                 seq=<suffix>[,...] legend=true|false
                                 legborder=true|false legopaque=true|false
                                 legseq=<suffix>[,...] legpos=<xfrac,yfrac>
                                 title=<value>
                                 auxmap=<map-name>|<color>-<color>[-<color>...]
                                 auxclip=<lo>,<hi> auxflip=true|false
                                 auxquant=<number>
                                 auxfunc=log|linear|histogram|histolog|sqrt|square|acos|cos
                                 auxmin=<number> auxmax=<number>
                                 auxlabel=<text> auxcrowd=<factor>
                                 auxwidth=<pixels> auxvisible=true|false
                                 forcebitmap=true|false compositor=0..1
                                 animate=<table> afmt=<in-format>
                                 astream=true|false acmd=<cmds>
                                 parallel=<int-value> xlog=true|false
                                 ylog=true|false xflip=true|false
                                 yflip=true|false xlabel=<text>
                                 ylabel=<text> aspect=<number>
                                 grid=true|false xcrowd=<number>
                                 ycrowd=<number> minor=true|false
                                 gridcolor=<rrggbb>|red|blue|...
                                 labelcolor=<rrggbb>|red|blue|...
                                 texttype=plain|antialias|latex
                                 fontsize=<int-value>
                                 fontstyle=standard|serif|mono
                                 fontweight=plain|bold|italic|bold_italic
                                 xmin=<number> xmax=<number> xsub=<lo>,<hi>
                                 ymin=<number> ymax=<number> ysub=<lo>,<hi>
                                 navaxes=xy|x|y xanchor=true|false
                                 yanchor=true|false zoomfactor=<number>
                                 leglabelN=<text>
                                 layerN=<layer-type> <layerN-specific-params>
```

If you don't have the stilts script installed, write "java -jar stilts.jar" instead of "stilts" - see Section 3. The available <stilts-flags> are listed in Section 2.1. For programmatic invocation, the Task class for this command is uk.ac.starlink.ttools.plot2.task.PlanePlot2Task.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

acmd = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the animation control table as specified by parameter animate, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be

separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`afmt = <in-format>` ***(String)***

Specifies the format of the animation control table as specified by parameter `animate`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`animate = <table>` ***(StarTable)***

If not null, this parameter causes the command to create a sequence of plots instead of just one. The parameter value is a table with one row for each frame to be produced. Columns in the table are interpreted as parameters which may take different values for each frame; the column name is the parameter name, and the value for a given frame is its value from that row. Animating like this is considerably more efficient than invoking the STILTS command in a loop.

The location of the animation control table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `afmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`aspect = <number>` ***(Double)***

Ratio of the unit length on the X axis to the unit length on the Y axis. If set to 1, the space will be isotropic. If not set (the default) the ratio will be determined by the given or calculated data bounds on both axes and the shape of the plotting region.

`astream = true|false` ***(Boolean)***

If set true, the animation control table specified by the `animate` parameter will be read as a stream. It is necessary to give the `afmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`auxclip = <lo>,<hi>` ***(Subrange)***

Defines a subrange of the colour ramp to be used for Aux shading. The value is specified as a (low,high) comma-separated pair of two numbers between 0 and 1.

If the full range `0,1` is used, the whole range of colours specified by the selected shader will be

used. But if for instance a value of `0,0.5` is given, only those colours at the left hand end of the ramp will be seen.

If the null (default) value is chosen, a default clip will be used. This generally covers most or all of the range 0-1 but for colour maps which fade to white, a small proportion of the lower end may be excluded, to ensure that all the colours are visually distinguishable from a white background. This default is usually a good idea if the colour map is being used with something like a scatter plot, where markers are plotted against a white background. However, for something like a density map when the whole plotting area is tiled with colours from the map, it may be better to supply the whole range `0,1` explicitly.

`auxcrowd = <factor>` ***(Double)***
Determines how closely the tick marks are spaced on the Aux axis, if visible. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

[Default: `1.0`]

`auxflip = true|false` ***(Boolean)***
If true, the colour map on the Aux axis will be reversed.

[Default: `false`]

`auxfunc = log|linear|histogram|histolog|sqrt|square|acos|cos` ***(Scaling)***
Defines the way that values in the Aux range are mapped to the selected colour ramp.

The available options are:

- `log`: Logarithmic scaling
- `linear`: Linear scaling
- `histogram`: Scaling follows data distribution, with linear axis
- `histolog`: Scaling follows data distribution, with logarithmic axis
- `sqrt`: Square root scaling
- `square`: Square scaling
- `acos`: Arccos Scaling
- `cos`: Cos Scaling

For all these options, the full range of data values is used, and displayed on the colour bar if applicable. The `Linear`, `Log`, `Square` and `Sqrt` options just apply the named function to the full data range. The histogram options on the other hand use a scaling function that corresponds to the actual distribution of the data, so that there are about the same number of points (or pixels, or whatever is being scaled) of each colour. The histogram options are somewhat more expensive, but can be a good choice if you are exploring data whose distribution is unknown or not well-behaved over its min-max range. The `Histogram` and `HistoLog` options both assign the colours in the same way, but they display the colour ramp with linear or logarithmic annotation respectively; the `HistoLog` option also ignores non-positive values.

[Default: `linear`]

`auxlabel = <text>` ***(String)***
Sets the label used to annotate the aux axis, if it is visible.

`auxmap = <map-name>|<color>-<color>[-<color>...]` ***(Shader)***
Color map used for Aux axis shading.

A mixed bag of colour ramps are available: `inferno`, `magma`, `plasma`, `viridis`, `cividis`, `cubehelix`, `sron`, `rainbow`, `rainbow2`, `rainbow3`, `pastel`, `accent`, `gnuplot`, `gnuplot2`, `specxby`, `set1`, `paired`, `hotcold`, `rdbu`, `piyg`, `brbg`, `cyan-magenta`, `red-blue`, `brg`, `heat`, `cold`, `light`, `greyscale`, `colour`, `standard`, `bugn`, `bupu`, `orrd`, `pubu`, `purd`, `painbow`, `huecl`, `hue`, `intensity`, `rgb_red`, `rgb_green`, `rgb_blue`, `hsv_h`, `hsv_s`, `hsv_v`, `yuv_y`, `yuv_u`, `yuv_v`, `scale_hsv_s`, `scale_hsv_v`, `scale_yuv_y`, `mask`, `blacker`, `whiter`, `transparency`. *Note:* many

of these, including rainbow-like ones, are frowned upon by the visualisation community.

You can also construct your own custom colour map by giving a sequence of colour names separated by minus sign ("`-`") characters. In this case the ramp is a linear interpolation between each pair of colours named, using the same syntax as when specifying a colour value. So for instance "`yellow-hotpink-#0000ff`" would shade from yellow via hot pink to blue.

[Default: `inferno`]

`auxmax = <number>` ***(Double)***
Maximum value of the data coordinate on the Aux axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`auxmin = <number>` ***(Double)***
Minimum value of the data coordinate on the Aux axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`auxquant = <number>` ***(Double)***
Allows the colour map used for the Aux axis to be quantised. If an integer value N is chosen then the colour map will be viewed as N discrete evenly-spaced levels, so that only N different colours will appear in the plot. This can be used to generate a contour-like effect, and may make it easier to trace the boundaries of regions of interest by eye.

If left blank, the colour map is nominally continuous (though in practice it may be quantised to a medium-sized number like 256).

`auxvisible = true|false` ***(Boolean)***
Determines whether the aux axis colour ramp is displayed alongside the plot.

If not supplied (the default), the aux axis will be visible when aux shading is used in any of the plotted layers.

`auxwidth = <pixels>` ***(Integer)***
Determines the lateral size of the aux colour ramp, if visible, in pixels.

[Default: `15`]

`compositor = 0..1` ***(Compositor)***
Defines how multiple overplotted partially transparent pixels are combined to form a resulting colour. The way this is used depends on the details of the specified plot.

Currently, this parameter takes a "boost" value in the range 0..1. If the value is zero, saturation semantics are used: RGB colours are added in proporition to their associated alpha value until the total alpha is saturated (reaches 1), after which additional pixels have no further effect. For larger boost values, the effect is similar, but any non-zero alpha in the output is boosted to the given minimum value. The effect of this is that even very slightly populated pixels can be visually distinguished from unpopulated ones which may not be the case for saturation composition.

[Default: `0.05`]

`fontsize = <int-value>` ***(Integer)***
Size of the text font in points.

[Default: `12`]

`fontstyle = standard|serif|mono` ***(FontType)***
Font style for text.

The available options are:

- `standard`
- `serif`
- `mono`

[Default: `standard`]

`fontweight = plain|bold|italic|bold_italic` ***(FontWeight)***
Font weight for text.

The available options are:

- `plain`
- `bold`
- `italic`
- `bold_italic`

[Default: `plain`]

`forcebitmap = true|false` ***(Boolean)***
Affects whether rendering of the data contents of a plot (though not axis labels etc) is always done to an intermediate bitmap rather than, where possible, being painted using graphics primitives. This is a rather arcane setting that may nevertheless have noticeable effects on the appearance and size of an output graphics file, as well as plotting time. For some types of plot (e.g. `shadingN=auto` or `shadingN=density`) it will have no effect, since this kind of rendering happens in any case.

When writing to vector graphics formats (PDF and PostScript), setting it true will force the data contents to be bitmapped. This may make the output less beautiful (round markers will no longer be perfectly round), but it may result in a much smaller file if there are very many data points.

When writing to bitmapped output formats (PNG, GIF, JPEG, ...), it fixes shapes to be the same as seen on the screen rather than be rendered at the mercy of the graphics system, which sometimes introduces small distortions.

[Default: `false`]

`grid = true|false` ***(Boolean)***
If true, grid lines are drawn on the plot at positions determined by the major tick marks. If false, they are absent.

[Default: `false`]

`gridcolor = <rrggbb>|red|blue|...` ***(Color)***
The color of the plot grid, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `light_grey`]

`insets = <top>,<left>,<bottom>,<right>` ***(Padding)***
Defines the amount of space in pixels around the actual plotting area. This space is used for axis labels, and other decorations and any left over forms an empty border.

The size and position of the actual plotting area is determined by this parameter along with `xpix` and `ypix`.

The value of this parameter is 4 comma separated integers: `<top>,<left>,<bottom>,<right>`. Any or all of these values may be left blank, in which case the corresponding margin will be calculated automatically according to how much space is required.

`labelcolor = <rrggbb>|red|blue|...` ***(Color)***

The color of axis labels and other plot annotations, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `black`]

`layerN = <layer-type> <layerN-specific-params>` ***(LayerType)***

Selects one of the available plot types for layerN. A plot consists of a plotting surface, set up using the various unsuffixed parameters of the plotting command, and zero or more plot layers. Each layer is introduced by a parameter with the name `layer<N>` where the suffix "`<N>`" is a label identifying the layer and is appended to all the parameter names which configure that layer. Suffixes may be any string, including the empty string.

This parameter may take one of the following values, described in more detail in Section 8.3:

- `mark (Section 8.3.1)`
- `size (Section 8.3.2)`
- `sizexy (Section 8.3.3)`
- `xyvector (Section 8.3.4)`
- `xyerror (Section 8.3.5)`
- `xyellipse (Section 8.3.6)`
- `xycorr (Section 8.3.7)`
- `link2 (Section 8.3.8)`
- `mark2 (Section 8.3.9)`
- `poly4 (Section 8.3.10)`
- `mark4 (Section 8.3.11)`
- `polygon (Section 8.3.12)`
- `area (Section 8.3.13)`
- `central (Section 8.3.14)`
- `lines (Section 8.3.15)`
- `marks (Section 8.3.16)`
- `yerrors (Section 8.3.17)`
- `xyerrors (Section 8.3.18)`
- `statline (Section 8.3.19)`
- `statmark (Section 8.3.20)`
- `arrayquantile (Section 8.3.21)`
- `line (Section 8.3.22)`
- `linearfit (Section 8.3.23)`
- `label (Section 8.3.24)`
- `arealabel (Section 8.3.25)`
- `contour (Section 8.3.26)`
- `grid (Section 8.3.27)`
- `fill (Section 8.3.28)`
- `quantile (Section 8.3.29)`
- `histogram (Section 8.3.30)`
- `kde (Section 8.3.31)`
- `knn (Section 8.3.32)`
- `densogram (Section 8.3.33)`

- `gaussian (Section 8.3.34)`
- `function (Section 8.3.35)`

Each of these layer types comes with a list of type-specific parameters to define the details of that layer, including some or all of the following groups:

- input table parameters (e.g. `inN`, `icmdN`)
- coordinate params referring to input table columns (e.g. `xN`, `yN`)
- layer style parameters (e.g. `shadingN`, `colorN`)

Every parameter notionally carries the same suffix `N`. However, if the suffix is not present, the application will try looking for a parameter with the same name with no suffix instead. In this way, if several layers have the same value for a given parameter (for instance input table), you can supply it using one unsuffixed parameter to save having to supply several parameters with the same value but different suffixes.

`legborder = true|false` ***(Boolean)***
If true, a line border is drawn around the legend.

[Default: `true`]

`legend = true|false` ***(Boolean)***
Whether to draw a legend or not. If no value is supplied, the decision is made automatically: a legend is drawn only if it would have more than one entry.

`leglabelN = <text>` ***(String)***
Sets the presentation label for the layer with a given suffix. This is the text which is displayed in the legend, if present. Multiple layers may use the same label, in which case they will be combined to form a single legend entry.

If no value is supplied (the default), the suffix itself is used as the label.

`legopaque = true|false` ***(Boolean)***
If true, the background of the legend is opaque, and the legend obscures any plot components behind it. Otherwise, it's transparent.

[Default: `true`]

`legpos = <xfrac,yfrac>` ***(double[])***
Determines the internal position of the legend on the plot. The value is a comma-separated pair of values giving the X and Y positions of the legend within the plotting bounds, so for instance "`0.5,0.5`" will put the legend right in the middle of the plot. If no value is supplied, the legend will appear outside the plot boundary.

`legseq = <suffix>[,...]` ***(String[])***
Determines which layers are represented in the legend (if present) and in which order they appear. The legend has a line for each layer label (as determined by the `leglabelN` parameter). If multiple layers have the same label, they will contribute to the same entry in the legend, with style icons plotted over each other. The value of this parameter is a comma-separated sequence of layer suffixes, which determines the order in which the legend entries appear. Layers with suffixes missing from this list do not show up in the legend at all.

If no value is supplied (the default), the sequence is the same as the layer plotting sequence (see `seq`).

`minor = true|false` ***(Boolean)***
If true, minor tick marks are painted along the axes as well as the major tick marks. Minor tick marks do not have associated grid lines.

[Default: `true`]

`navaxes = xy|x|y` ***(boolean[])***
Determines the axes which are affected by the interactive navigation actions (pan and zoom).

The default is `xy`, which means that the various mouse gestures will provide panning and zooming in both X and Y directions. However, if it is set to (for instance) `x` then the mouse will only allow panning and zooming in the horizontal direction, with the vertical extent fixed.

[Default: `xy`]

`omode = swing|out|cgi|discard|auto` ***(PaintMode)***

Determines how the drawn plot will be output, see Section 8.5.

- `swing`: Plot will be displayed in a window on the screen. This plot is "live"; it can be resized and (except for old-style plots) navigated around with mouse actions in the same way as plots in TOPCAT.
- `out`: Plot will be written to a file given by `out` using the graphics format given by `ofmt`.
- `cgi`: Plot will be written in a way suitable for CGI use direct from a web server. The output is in the graphics format given by `ofmt`, preceded by a suitable "Content-type" declaration.
- `discard`: Plot is drawn, but discarded. There is no output.
- `auto`: Behaves as `swing` or `out` mode depending on presence of `out` parameter

[Default: `auto`]

`parallel = <int-value>` ***(Integer)***

Determines how many threads will run in parallel if animation output is being produced. Only used if the `animate` parameter is supplied. The default value is the number of processors apparently available to the JVM.

[Default: `20`]

`seq = <suffix>[,...]` ***(String[])***

Contains a comma-separated list of layer suffixes to determine the order in which layers are drawn on the plot. This can affect which symbol are plotted on top of, and so potentially obscure, which other ones.

When specifying a plot, multiple layers may be specified, each introduced by a parameter `layer<N>`, where `<N>` is a different (arbitrary) suffix labelling the layer, and is appended to all the parameters specific to defining that layer.

By default the layers are drawn on the plot in the order in which the `layer*` parameters appear on the command line. However if this parameter is specified, each comma-separated element is interpreted as a layer suffix, giving the ordered list of layers to plot. Every element of the list must be a suffix with a corresponding `layer` parameter, but missing or repeated elements are allowed.

`storage = simple|memory|disk|policy|cache|basic-cache|persistent|parallel` ***(DataStoreFactory)***

Determines the way that data is accessed when constructing the plot. There are two main options, cached or not. If no caching is used then rows are read sequentially from the specified input table(s) every time they are required. This generally requires a small resource footprint (though that can depend on how the table is specified) and makes sense if the data only needs to be scanned once or perhaps if the table is very large. If caching is used then the required data is read once from the specified input table(s), then prepared and cached before any plotting is performed, and plots are done using this cached data. This may use a significant amount of storage for large tables but it's usually more sensible (faster) if the data will need to be scanned multiple times. There are various options for cache storage.

The options are:

- `simple`: no caching, data read directly from input table
- `memory`: cached to memory; OutOfMemoryError possible for very large plots
- `disk`: cached to disk
- `policy`: cached using application-wide default storage policy, which is usually *adaptive*

(memory/disk hybrid)
- `persistent`: cached to persistent files on disk, in the system temporary directory (defined by system property `java.io.tmpdir`). If this is used, plot data will be stored on disk in a way that means they can be re-used between STILTS invocations, so data preparation can be avoided on subsequent runs. Note however it can leave potentially large files in your temporary directory.
- `cache`: synonym for `memory` (backward compatibility)
- `basic-cache`: dumber version of `memory` (no optimisation for constant-valued columns)
- `parallel`: experimental version of memory-based cache that reads into the cache in parallel for large files. This will make the plot faster to prepare, but interaction is a bit slower and sequence-dependent attributes of the plot may not come out right. This experimental option may be withdrawn or modified in future releases.

The default value is `memory` if a live plot is being generated (`omode=swing`), since in that case the plot needs to be redrawn every time the user performs plot navigation actions or resizes the window, or if animations are being produced. Otherwise (e.g. output to a graphics file) the default is `simple`.

[Default: `simple`]

`texttype = plain|antialias|latex` ***(TextSyntax)***

Determines how to turn label text into characters on the plot. `Plain` and `Antialias` both take the text at face value, but `Antialias` smooths the characters. `LaTeX` interprets the text as LaTeX source code and typesets it accordingly.

When not using LaTeX, antialiased text usually looks nicer, but can be perceptibly slower to plot. At time of writing, on MacOS antialiased text seems to be required to stop the writing coming out upside-down for non-horizontal text (MacOS java bug).

[Default: `plain`]

`title = <value>` ***(String)***

Text of a title to be displayed at the top of the plot. If null, the default, no title is shown and there's more space for the graphics.

`xanchor = true|false` ***(Boolean)***

If true, then zoom actions will work in such a way that the zero point on the X axis stays in the same position on the plot.

[Default: `false`]

`xcrowd = <number>` ***(Double)***

Determines how closely the tick marks are spaced on the X axis. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

[Default: `1`]

`xflip = true|false` ***(Boolean)***

If true, the scale on the X axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

`xlabel = <text>` ***(String)***

Gives a label to be used for annotating axis X A default value based on the plotted data will be used if no value is supplied.

[Default: `X`]

`xlog = true|false` ***(Boolean)***

If false (the default), the scale on the X axis is linear, if true it is logarithmic.

[Default: `false`]

`xmax = <number>` ***(Double)***
Maximum value of the data coordinate on the X axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`xmin = <number>` ***(Double)***
Minimum value of the data coordinate on the X axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`xpix = <int-value>` ***(Integer)***
Size of the output image in the X direction in pixels. This includes space for any axis labels, padding and other decoration outside the plot area itself. See also `insets`.

[Default: `500`]

`xsub = <lo>,<hi>` ***(Subrange)***
Defines a normalised adjustment to the data range of the X axis. The value may be specified as a comma-separated pair of two numbers, giving the lower and upper bounds of the range of of interest respectively. This sub-range is applied to the data range that would otherwise be used, either automatically calculated or explicitly supplied; zero corresponds to the lower bound and one to the upper.

The default value "`0,1`" therefore has no effect. The range could be restricted to its lower half with the value `0,0.5`.

[Default: `0,1`]

`yanchor = true|false` ***(Boolean)***
If true, then zoom actions will work in such a way that the zero point on the Y axis stays in the same position on the plot.

[Default: `false`]

`ycrowd = <number>` ***(Double)***
Determines how closely the tick marks are spaced on the Y axis. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

[Default: `1`]

`yflip = true|false` ***(Boolean)***
If true, the scale on the Y axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

`ylabel = <text>` ***(String)***
Gives a label to be used for annotating axis Y A default value based on the plotted data will be used if no value is supplied.

[Default: `Y`]

`ylog = true|false` ***(Boolean)***
If false (the default), the scale on the Y axis is linear, if true it is logarithmic.

[Default: `false`]

`ymax = <number>` ***(Double)***
Maximum value of the data coordinate on the Y axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`ymin = <number>` ***(Double)***
Minimum value of the data coordinate on the Y axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

ypix = <int-value> ***(Integer)***

Size of the output image in the Y direction in pixels. This includes space for any axis labels, padding and other decoration outside the plot area itself. See also `insets`.

[Default: `400`]

ysub = <lo>,<hi> ***(Subrange)***

Defines a normalised adjustment to the data range of the Y axis. The value may be specified as a comma-separated pair of two numbers, giving the lower and upper bounds of the range of of interest respectively. This sub-range is applied to the data range that would otherwise be used, either automatically calculated or explicitly supplied; zero corresponds to the lower bound and one to the upper.

The default value "`0,1`" therefore has no effect. The range could be restricted to its lower half with the value `0,0.5`.

[Default: `0,1`]

zoomfactor = <number> ***(Double)***

Sets the amount by which the plot view zooms in or out for each unit of mouse wheel movement. A value of 1 means that mouse wheel zooming has no effect. A higher value means that the mouse wheel zooms faster and a value nearer 1 means it zooms slower. Values below 1 are not permitted.

[Default: `1.2`]

B.10.2 Examples

Here are some examples of `plot2plane`:

```
stilts plot2plane yflip=true layer_1=mark in_1=cat.fits x_1=BMAG-RMAG y_1=BMAG
```

This is a colour-magnitude diagram where the input table has columns named RMAG and BMAG. The Y axis is inverted so that the magnitude values increase downwards not up. The plot is displayed in a window on the screen, and may be panned and zoomed with the mouse.

```
stilts plot2plane layer=histogram in=hip_main.fits x=plx xlog=true
                  xlabel=Parallax ylabel=
```

Plots a histogram of parallaxes for Hipparcos data, with a logarithmic X axis. The axes are labelled explicitly, with an empty string in the case of the Y axis.

```
stilts plot2plane xpix=600 ypix=500
                  in=gavo_g2.fits x=X y=Y
                  shading=aux aux='atan2(vely,velx)' auxmap=hue auxvisible=false
                  layer_m=mark shape_m=cross size_m=4
                  layer_v=xyvector xdelta_v=velx ydelta_v=vely scale_v=2
                  out=velocities.pdf
```

Two layers are plotted, point markers representing position (4 pixels radius, shaped like crosses) and vectors representing velocity. Both markers and vectors are coloured according to the direction (arctan(vely/velx)) of the arrows, so it's easy to see points moving in similar directions; the "hue" colour map is good for this, since it's periodic, so values of +Pi and -Pi have the same colour. Since it's not very revealing in this case, display of the aux axis colour ramp beside the plot has been turned off. Since the X and Y coordinates and the colouring is common to both layers, the relevant parameters can given without suffixes to avoid having to repeat them. Output is to a PDF file.

```
stilts plot2plane xmin=0 xmax=6.283 ymin=-1 ymax=1 xlabel=Time
                  layer=function axis=horizontal xname=time fexpr='sin(time)'
```

```
                dash=3,2 thick=4 color=ee6aa7
```

Plots a sine curve to the screen. Initially the view is of one period, but you can pan and zoom interactively to see any range. The line is plotted in hot pink, four pixels wide, with a custom dash pattern. Since the function layer type has no data coordinates, no input table is required. The layer suffix here is the empty string; since there's only one layer, it doesn't cause any problems.

```
stilts plot2plane ylog=true xflip=true xmin=-5.2 xmax=3.8 ymin=250 ymax=3.5e5
           in1=6dfgs_E7.fits x1=bmag-rmag y1=vel
           layer1a=mark color1a=cyan
           layer1b=contour color1b=yellow smooth1b=9 scaling1b=log
           layer1c=mark icmd1c='every 35;select star'
                        shape1c=filled_triangle_down size1c=5 color1c=red
                        shading1c=transparent opaque1c=3
           layer2=function fexpr2='exp(x*2+12)' color2=black antialias2=true
                           dash2=dash thick2=3
           leglabel1a=Population leglabel1c=Sample legpos=.95,.95 legseq=1a,1c
           fontsize=16 texttype=latex ylabel="v\,/\,km.s^{-1}" xlabel=colour
```

There are four layers: 1a, 1b and 1c use the same positional data from the same input file, so the positional coordinates common to them are given the suffix "1". Layer "2" is unrelated, and has no input data, since it's just an analytic function. The legend is positioned to taste, and its content is manipulated so that only datasets 1a and 1c are described, and they are given custom names (the default would be their suffix names).

B.11 `plot2sky`: Draws a sky plot

`plot2sky` draws plots on the celestial sphere. This can be represented in a number of ways, controlled by the `projection` parameter; by default the view is of a rotatable sphere seen from the outside (which approximates to a tangent projection for small regions of the sky), but Hammer-Aitoff and Plate Carée projections are also available. A number of options are also provided for drawing and labelling the grid showing celestial coordinates.

Positional coordinates are specified as `lon`, `lat` pairs giving longitude and latitude in decimal degrees. By default these are represented in the output in the same, unlabelled, coordinate system. However the command can can also transform between different coordinate systems if you specify the data and view systems e.g.:

```
plot2sky viewsys=galactic
         layer1=mark in1=cat.fits lon1=RA2000 lat1=DEC2000 datasys1=equatorial
```

Content is added to the plot by specifying one or more *plot layers* using the `layerN` parameter. The `N` part is a suffix applied to all the parameters affecting a given layer; any suffix (including the empty string) may be used. Available layers for this plot type are: `mark` (Section 8.3.1), `size` (Section 8.3.2), `sizexy` (Section 8.3.3), `skyvector` (Section 8.3.36), `skyellipse` (Section 8.3.37), `skycorr` (Section 8.3.38), `link2` (Section 8.3.8), `mark2` (Section 8.3.9), `poly4` (Section 8.3.10), `mark4` (Section 8.3.11), `polygon` (Section 8.3.12), `area` (Section 8.3.13), `central` (Section 8.3.14), `label` (Section 8.3.24), `arealabel` (Section 8.3.25), `contour` (Section 8.3.26), `skydensity` (Section 8.3.39), `healpix` (Section 8.3.40), `skygrid` (Section 8.3.41).

B.11.1 Usage

The usage of `plot2sky` is

```
stilts <stilts-flags> plot2sky xpix=<int-value> ypix=<int-value>
                              insets=<top>,<left>,<bottom>,<right>
                              omode=swing|out|cgi|discard|auto
```

```
storage=simple|memory|disk|policy|cache|basic-cache|persiste
seq=<suffix>[,...] legend=true|false
legborder=true|false legopaque=true|false
legseq=<suffix>[,...] legpos=<xfrac,yfrac>
title=<value>
auxmap=<map-name>|<color>-<color>[-<color>...]
auxclip=<lo>,<hi> auxflip=true|false
auxquant=<number>
auxfunc=log|linear|histogram|histolog|sqrt|square|acos|cos
auxmin=<number> auxmax=<number>
auxlabel=<text> auxcrowd=<factor>
auxwidth=<pixels> auxvisible=true|false
forcebitmap=true|false compositor=0..1
animate=<table> afmt=<in-format>
astream=true|false acmd=<cmds>
parallel=<int-value>
projection=sin|aitoff|aitoff0|car|car0
viewsys=equatorial|galactic|supergalactic|ecliptic
reflectlon=true|false grid=true|false
scalebar=true|false
labelpos=Auto|External|Internal|Basic|Hybrid|None
sex=true|false crowd=<number>
gridcolor=<rrggbb>|red|blue|...
labelcolor=<rrggbb>|red|blue|...
gridaa=true|false
texttype=plain|antialias|latex
fontsize=<int-value>
fontstyle=standard|serif|mono
fontweight=plain|bold|italic|bold_italic
clon=<degrees> clat=<degrees>
radius=<degrees> zoomfactor=<number>
leglabelN=<text>
layerN=<layer-type> <layerN-specific-params>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.plot2.task.SkyPlot2Task`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`acmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the animation control table as specified by parameter `animate`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`afmt = <in-format>` ***(String)***

Specifies the format of the animation control table as specified by parameter `animate`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`animate = <table>` ***(StarTable)***

If not null, this parameter causes the command to create a sequence of plots instead of just one. The parameter value is a table with one row for each frame to be produced. Columns in the table are interpreted as parameters which may take different values for each frame; the column name is the parameter name, and the value for a given frame is its value from that row. Animating like this is considerably more efficient than invoking the STILTS command in a loop.

The location of the animation control table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `afmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`astream = true|false` ***(Boolean)***

If set true, the animation control table specified by the `animate` parameter will be read as a stream. It is necessary to give the `afmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`auxclip = <lo>,<hi>` ***(Subrange)***

Defines a subrange of the colour ramp to be used for Aux shading. The value is specified as a (low,high) comma-separated pair of two numbers between 0 and 1.

If the full range `0,1` is used, the whole range of colours specified by the selected shader will be used. But if for instance a value of `0,0.5` is given, only those colours at the left hand end of the ramp will be seen.

If the null (default) value is chosen, a default clip will be used. This generally covers most or all of the range 0-1 but for colour maps which fade to white, a small proportion of the lower end may be excluded, to ensure that all the colours are visually distinguishable from a white background. This default is usually a good idea if the colour map is being used with something like a scatter plot, where markers are plotted against a white background. However, for something like a density map when the whole plotting area is tiled with colours from the map, it may be better to supply the whole range `0,1` explicitly.

`auxcrowd = <factor>` ***(Double)***

Determines how closely the tick marks are spaced on the Aux axis, if visible. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

[Default: `1.0`]

`auxflip = true|false` ***(Boolean)***

If true, the colour map on the Aux axis will be reversed.

[Default: `false`]

`auxfunc = log|linear|histogram|histolog|sqrt|square|acos|cos` ***(Scaling)***

Defines the way that values in the Aux range are mapped to the selected colour ramp.

The available options are:

- `log`: Logarithmic scaling
- `linear`: Linear scaling
- `histogram`: Scaling follows data distribution, with linear axis
- `histolog`: Scaling follows data distribution, with logarithmic axis
- `sqrt`: Square root scaling
- `square`: Square scaling
- `acos`: Arccos Scaling
- `cos`: Cos Scaling

For all these options, the full range of data values is used, and displayed on the colour bar if applicable. The `Linear`, `Log`, `Square` and `Sqrt` options just apply the named function to the full data range. The histogram options on the other hand use a scaling function that corresponds to the actual distribution of the data, so that there are about the same number of points (or pixels, or whatever is being scaled) of each colour. The histogram options are somewhat more expensive, but can be a good choice if you are exploring data whose distribution is unknown or not well-behaved over its min-max range. The `Histogram` and `HistoLog` options both assign the colours in the same way, but they display the colour ramp with linear or logarithmic annotation respectively; the `HistoLog` option also ignores non-positive values.

[Default: `linear`]

`auxlabel = <text>` ***(String)***
Sets the label used to annotate the aux axis, if it is visible.

`auxmap = <map-name>|<color>-<color>[-<color>...]` ***(Shader)***
Color map used for Aux axis shading.

A mixed bag of colour ramps are available: `inferno`, `magma`, `plasma`, `viridis`, `cividis`, `cubehelix`, `sron`, `rainbow`, `rainbow2`, `rainbow3`, `pastel`, `accent`, `gnuplot`, `gnuplot2`, `specxby`, `set1`, `paired`, `hotcold`, `rdbu`, `piyg`, `brbg`, `cyan-magenta`, `red-blue`, `brg`, `heat`, `cold`, `light`, `greyscale`, `colour`, `standard`, `bugn`, `bupu`, `orrd`, `pubu`, `purd`, `painbow`, `huecl`, `hue`, `intensity`, `rgb_red`, `rgb_green`, `rgb_blue`, `hsv_h`, `hsv_s`, `hsv_v`, `yuv_y`, `yuv_u`, `yuv_v`, `scale_hsv_s`, `scale_hsv_v`, `scale_yuv_y`, `mask`, `blacker`, `whiter`, `transparency`. *Note:* many of these, including rainbow-like ones, are frowned upon by the visualisation community.

You can also construct your own custom colour map by giving a sequence of colour names separated by minus sign ("`-`") characters. In this case the ramp is a linear interpolation between each pair of colours named, using the same syntax as when specifying a colour value. So for instance "`yellow-hotpink-#0000ff`" would shade from yellow via hot pink to blue.

[Default: `inferno`]

`auxmax = <number>` ***(Double)***
Maximum value of the data coordinate on the Aux axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`auxmin = <number>` ***(Double)***
Minimum value of the data coordinate on the Aux axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`auxquant = <number>` ***(Double)***
Allows the colour map used for the Aux axis to be quantised. If an integer value N is chosen then the colour map will be viewed as N discrete evenly-spaced levels, so that only N different colours will appear in the plot. This can be used to generate a contour-like effect, and may make it easier to trace the boundaries of regions of interest by eye.

If left blank, the colour map is nominally continuous (though in practice it may be quantised to a medium-sized number like 256).

auxvisible = true|false ***(Boolean)***
Determines whether the aux axis colour ramp is displayed alongside the plot.

If not supplied (the default), the aux axis will be visible when aux shading is used in any of the plotted layers.

auxwidth = <pixels> ***(Integer)***
Determines the lateral size of the aux colour ramp, if visible, in pixels.

[Default: `15`]

clat = <degrees> ***(Double)***
Latitude of the central position of the plot in decimal degrees. Use with `clon` and `radius`. If the center is not specified, the field of view is determined from the data.

clon = <degrees> ***(Double)***
Longitude of the central position of the plot in decimal degrees. Use with `clat` and `radius`. If the center is not specified, the field of view is determined from the data.

compositor = 0..1 ***(Compositor)***
Defines how multiple overplotted partially transparent pixels are combined to form a resulting colour. The way this is used depends on the details of the specified plot.

Currently, this parameter takes a "boost" value in the range 0..1. If the value is zero, saturation semantics are used: RGB colours are added in proporition to their associated alpha value until the total alpha is saturated (reaches 1), after which additional pixels have no further effect. For larger boost values, the effect is similar, but any non-zero alpha in the output is boosted to the given minimum value. The effect of this is that even very slightly populated pixels can be visually distinguished from unpopulated ones which may not be the case for saturation composition.

[Default: `0.05`]

crowd = <number> ***(Double)***
Determines how closely sky grid lines are spaced. The default value is 1, meaning normal crowding. Larger values result in more grid lines, and smaller values in fewer grid lines.

[Default: `1`]

fontsize = <int-value> ***(Integer)***
Size of the text font in points.

[Default: `12`]

fontstyle = standard|serif|mono ***(FontType)***
Font style for text.

The available options are:

- `standard`
- `serif`
- `mono`

[Default: `standard`]

fontweight = plain|bold|italic|bold_italic ***(FontWeight)***
Font weight for text.

The available options are:

- `plain`
- `bold`
- `italic`
- `bold_italic`

[Default: `plain`]

`forcebitmap = true|false` ***(Boolean)***
Affects whether rendering of the data contents of a plot (though not axis labels etc) is always done to an intermediate bitmap rather than, where possible, being painted using graphics primitives. This is a rather arcane setting that may nevertheless have noticeable effects on the appearance and size of an output graphics file, as well as plotting time. For some types of plot (e.g. `shadingN=auto` or `shadingN=density`) it will have no effect, since this kind of rendering happens in any case.

When writing to vector graphics formats (PDF and PostScript), setting it true will force the data contents to be bitmapped. This may make the output less beautiful (round markers will no longer be perfectly round), but it may result in a much smaller file if there are very many data points.

When writing to bitmapped output formats (PNG, GIF, JPEG, ...), it fixes shapes to be the same as seen on the screen rather than be rendered at the mercy of the graphics system, which sometimes introduces small distortions.

[Default: `false`]

`grid = true|false` ***(Boolean)***
If true, sky coordinate grid lines are drawn on the plot. If false, they are absent.

[Default: `true`]

`gridaa = true|false` ***(Boolean)***
If true, grid lines are drawn with antialiasing. Antialiased lines look smoother, but may take perceptibly longer to draw. Only has any effect for bitmapped output formats.

[Default: `false`]

`gridcolor = <rrggbb>|red|blue|...` ***(Color)***
The color of the plot grid, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `light_grey`]

`insets = <top>,<left>,<bottom>,<right>` ***(Padding)***
Defines the amount of space in pixels around the actual plotting area. This space is used for axis labels, and other decorations and any left over forms an empty border.

The size and position of the actual plotting area is determined by this parameter along with `xpix` and `ypix`.

The value of this parameter is 4 comma separated integers: `<top>,<left>,<bottom>,<right>`. Any or all of these values may be left blank, in which case the corresponding margin will be calculated automatically according to how much space is required.

`labelcolor = <rrggbb>|red|blue|...` ***(Color)***
The color of axis labels and other plot annotations, given by name or as a hexadecimal RGB value.

The standard plotting colour names are `red`, `blue`, `green`, `grey`, `magenta`, `cyan`, `orange`, `pink`, `yellow`, `black`, `light_grey`, `white`. However, many other common colour names (too many to

list here) are also understood. The list currently contains those colour names understood by most web browsers, from `AliceBlue` to `YellowGreen`, listed e.g. in the *Extended color keywords* section of the CSS3 standard.

Alternatively, a six-digit hexadecimal number *RRGGBB* may be supplied, optionally prefixed by "`#`" or "`0x`", giving red, green and blue intensities, e.g. "`ff00ff`", "`#ff00ff`" or "`0xff00ff`" for magenta.

[Default: `black`]

`labelpos = Auto|External|Internal|Basic|Hybrid|None` ***(SkyAxisLabeller)***

Controls whether and where the numeric annotations of the lon/lat axes are displayed. The default option `Auto` usually does the sensible thing, but other options exist to force labelling internally or externally to the plot region, or to remove numeric labels altogether.

The available options are:

- `Auto`: Uses `External` or `Internal` policy according to whether the sky fills the plot bounds or not
- `External`: Labels are drawn outside the plot bounds
- `Internal`: Labels are drawn inside the plot bounds
- `Basic`: Labels are drawn somewhere near the grid line
- `Hybrid`: Grid lines are labelled outside the plot bounds where possible, but inside if they would otherwise be invisible
- `None`: Axes are not labelled

[Default: `Auto`]

`layerN = <layer-type> <layerN-specific-params>` ***(LayerType)***

Selects one of the available plot types for layerN. A plot consists of a plotting surface, set up using the various unsuffixed parameters of the plotting command, and zero or more plot layers. Each layer is introduced by a parameter with the name `layer<N>` where the suffix "`<N>`" is a label identifying the layer and is appended to all the parameter names which configure that layer. Suffixes may be any string, including the empty string.

This parameter may take one of the following values, described in more detail in Section 8.3:

- `mark (Section 8.3.1)`
- `size (Section 8.3.2)`
- `sizexy (Section 8.3.3)`
- `skyvector (Section 8.3.36)`
- `skyellipse (Section 8.3.37)`
- `skycorr (Section 8.3.38)`
- `link2 (Section 8.3.8)`
- `mark2 (Section 8.3.9)`
- `poly4 (Section 8.3.10)`
- `mark4 (Section 8.3.11)`
- `polygon (Section 8.3.12)`
- `area (Section 8.3.13)`
- `central (Section 8.3.14)`
- `label (Section 8.3.24)`
- `arealabel (Section 8.3.25)`
- `contour (Section 8.3.26)`
- `skydensity (Section 8.3.39)`
- `healpix (Section 8.3.40)`
- `skygrid (Section 8.3.41)`

Each of these layer types comes with a list of type-specific parameters to define the details of that layer, including some or all of the following groups:

- input table parameters (e.g. `inN`, `icmdN`)
- coordinate params referring to input table columns (e.g. `xN`, `yN`)
- layer style parameters (e.g. `shadingN`, `colorN`)

Every parameter notionally carries the same suffix `N`. However, if the suffix is not present, the application will try looking for a parameter with the same name with no suffix instead. In this way, if several layers have the same value for a given parameter (for instance input table), you can supply it using one unsuffixed parameter to save having to supply several parameters with the same value but different suffixes.

`legborder = true|false` ***(Boolean)***
If true, a line border is drawn around the legend.

[Default: `true`]

`legend = true|false` ***(Boolean)***
Whether to draw a legend or not. If no value is supplied, the decision is made automatically: a legend is drawn only if it would have more than one entry.

`leglabelN = <text>` ***(String)***
Sets the presentation label for the layer with a given suffix. This is the text which is displayed in the legend, if present. Multiple layers may use the same label, in which case they will be combined to form a single legend entry.

If no value is supplied (the default), the suffix itself is used as the label.

`legopaque = true|false` ***(Boolean)***
If true, the background of the legend is opaque, and the legend obscures any plot components behind it. Otherwise, it's transparent.

[Default: `true`]

`legpos = <xfrac,yfrac>` ***(double[])***
Determines the internal position of the legend on the plot. The value is a comma-separated pair of values giving the X and Y positions of the legend within the plotting bounds, so for instance "`0.5,0.5`" will put the legend right in the middle of the plot. If no value is supplied, the legend will appear outside the plot boundary.

`legseq = <suffix>[,...]` ***(String[])***
Determines which layers are represented in the legend (if present) and in which order they appear. The legend has a line for each layer label (as determined by the `leglabelN` parameter). If multiple layers have the same label, they will contribute to the same entry in the legend, with style icons plotted over each other. The value of this parameter is a comma-separated sequence of layer suffixes, which determines the order in which the legend entries appear. Layers with suffixes missing from this list do not show up in the legend at all.

If no value is supplied (the default), the sequence is the same as the layer plotting sequence (see `seq`).

`omode = swing|out|cgi|discard|auto` ***(PaintMode)***
Determines how the drawn plot will be output, see Section 8.5.

- `swing`: Plot will be displayed in a window on the screen. This plot is "live"; it can be resized and (except for old-style plots) navigated around with mouse actions in the same way as plots in TOPCAT.
- `out`: Plot will be written to a file given by `out` using the graphics format given by `ofmt`.
- `cgi`: Plot will be written in a way suitable for CGI use direct from a web server. The output is in the graphics format given by `ofmt`, preceded by a suitable "Content-type" declaration.
- `discard`: Plot is drawn, but discarded. There is no output.
- `auto`: Behaves as `swing` or `out` mode depending on presence of `out` parameter

[Default: `auto`]

`parallel = <int-value>` ***(Integer)***
Determines how many threads will run in parallel if animation output is being produced. Only used if the `animate` parameter is supplied. The default value is the number of processors apparently available to the JVM.

[Default: `20`]

`projection = sin|aitoff|aitoff0|car|car0` ***(Projection)***
Sky projection used to display the plot.

The available options are:

- `sin`: rotatable sphere
- `aitoff`: Hammer-Aitoff projection with lon=0 at center
- `aitoff0`: Hammer-Aitoff projection with lon=180 at center
- `car`: Plate Carree projection (lon/lat on Cartesian axes) with lon=0 at center
- `car0`: Plate Carree Projection (lon/lat on Cartesian axes) with lon=180 at center

Note that for historical reasons the projections named "`aitoff`" denote the equal-area Hammer-Aitoff projection and not the Aitoff projection itself. The Hammer-Aitoff projection is defined as:

- `x = [2sqrt(2)/sqrt(1+cos(lat)cos(lon/2))]cos(lat)sin(lon/2)`
- `y = [sqrt(2)/sqrt(1+cos(lat)cos(lon/2))]sin(lat)`

[Default: `sin`]

`radius = <degrees>` ***(Double)***
Approximate radius of the plot field of view in degrees. Only used if `clon` and `clat` are also specified.

[Default: `1`]

`reflectlon = true|false` ***(Boolean)***
Whether to invert the celestial sphere by displaying the longitude axis increasing right-to-left rather than left-to-right. It is conventional to display the celestial sphere in this way because that's what it looks like from the earth, so the default is `true`. Set it false to see the sphere from the outside.

[Default: `true`]

`scalebar = true|false` ***(Boolean)***
If true, a small bar is drawn near the bottom left of the plot annotated with a distance in degrees, arc minutes or arc seconds, to make it easier to determine the size of features on the plot by eye.

[Default: `true`]

`seq = <suffix>[,...]` ***(String[])***
Contains a comma-separated list of layer suffixes to determine the order in which layers are drawn on the plot. This can affect which symbol are plotted on top of, and so potentially obscure, which other ones.

When specifying a plot, multiple layers may be specified, each introduced by a parameter `layer<N>`, where `<N>` is a different (arbitrary) suffix labelling the layer, and is appended to all the parameters specific to defining that layer.

By default the layers are drawn on the plot in the order in which the `layer*` parameters appear on the command line. However if this parameter is specified, each comma-separated element is interpreted as a layer suffix, giving the ordered list of layers to plot. Every element of the list must be a suffix with a corresponding `layer` parameter, but missing or repeated elements are

allowed.

`sex = true|false` ***(Boolean)***

If true, grid line labels are written in sexagesimal notation, if false in decimal degrees.

[Default: `true`]

`storage = simple|memory|disk|policy|cache|basic-cache|persistent|parallel` ***(DataStoreFactory)***

Determines the way that data is accessed when constructing the plot. There are two main options, cached or not. If no caching is used then rows are read sequentially from the specified input table(s) every time they are required. This generally requires a small resource footprint (though that can depend on how the table is specified) and makes sense if the data only needs to be scanned once or perhaps if the table is very large. If caching is used then the required data is read once from the specified input table(s), then prepared and cached before any plotting is performed, and plots are done using this cached data. This may use a significant amount of storage for large tables but it's usually more sensible (faster) if the data will need to be scanned multiple times. There are various options for cache storage.

The options are:

- `simple`: no caching, data read directly from input table
- `memory`: cached to memory; OutOfMemoryError possible for very large plots
- `disk`: cached to disk
- `policy`: cached using application-wide default storage policy, which is usually *adaptive* (memory/disk hybrid)
- `persistent`: cached to persistent files on disk, in the system temporary directory (defined by system property `java.io.tmpdir`). If this is used, plot data will be stored on disk in a way that means they can be re-used between STILTS invocations, so data preparation can be avoided on subsequent runs. Note however it can leave potentially large files in your temporary directory.
- `cache`: synonym for `memory` (backward compatibility)
- `basic-cache`: dumber version of `memory` (no optimisation for constant-valued columns)
- `parallel`: experimental version of memory-based cache that reads into the cache in parallel for large files. This will make the plot faster to prepare, but interaction is a bit slower and sequence-dependent attributes of the plot may not come out right. This experimental option may be withdrawn or modified in future releases.

The default value is `memory` if a live plot is being generated (`omode=swing`), since in that case the plot needs to be redrawn every time the user performs plot navigation actions or resizes the window, or if animations are being produced. Otherwise (e.g. output to a graphics file) the default is `simple`.

[Default: `simple`]

`texttype = plain|antialias|latex` ***(TextSyntax)***

Determines how to turn label text into characters on the plot. `Plain` and `Antialias` both take the text at face value, but `Antialias` smooths the characters. `LaTeX` interprets the text as LaTeX source code and typesets it accordingly.

When not using LaTeX, antialiased text usually looks nicer, but can be perceptibly slower to plot. At time of writing, on MacOS antialiased text seems to be required to stop the writing coming out upside-down for non-horizontal text (MacOS java bug).

[Default: `plain`]

`title = <value>` ***(String)***

Text of a title to be displayed at the top of the plot. If null, the default, no title is shown and there's more space for the graphics.

`viewsys = equatorial|galactic|supergalactic|ecliptic` ***(SkySys)***

The sky coordinate system used for the generated plot.

Choice of this value goes along with the data coordinate system that may be specified for plot layers. If unspecified, a generic longitude/latitude system is used, and all lon/lat coordinates in the plotted data layers are assumed to be in the same system. If a value is supplied for this parameter, then a sky system must (implicitly or explicitly) be supplied for each data layer, and the coordinates are converted from data to view system before being plotted.

The available options are:

- `equatorial`: J2000 equatorial system
- `galactic`: IAU 1958 galactic system
- `supergalactic`: De Vaucouleurs supergalactic system
- `ecliptic`: ecliptic system based on conversion at 2000.0

`xpix = <int-value>` ***(Integer)***
Size of the output image in the X direction in pixels. This includes space for any axis labels, padding and other decoration outside the plot area itself. See also `insets`.

[Default: `500`]

`ypix = <int-value>` ***(Integer)***
Size of the output image in the Y direction in pixels. This includes space for any axis labels, padding and other decoration outside the plot area itself. See also `insets`.

[Default: `400`]

`zoomfactor = <number>` ***(Double)***
Sets the amount by which the plot view zooms in or out for each unit of mouse wheel movement. A value of 1 means that mouse wheel zooming has no effect. A higher value means that the mouse wheel zooms faster and a value nearer 1 means it zooms slower. Values below 1 are not permitted.

[Default: `1.2`]

B.11.2 Examples

Here are some examples of `plot2sky`:

```
stilts plot2sky in=messier.xml lon=RA lat=DEC
               layer.pos=mark size.pos=4
               layer.txt=label label.txt=Name color.txt=slategrey
```

Plots the positions of all the Messier objects on the sky, with text labels giving their object names. This displays a sphere on the screen that you can rotate/zoom using the mouse.

```
stilts plot2sky projection=aitoff
               xpix=600 ypix=300
               gridcolour=green labelcolour=black
               fontsize=10 gridaa=true texttype=antialias
               sex=true crowd=4
```

This just plots a celestial coordinate grid with no data. Various options are tweaked to adjust the appearance of the grid.

```
stilts plot2sky xpix=1000 ypix=500 fontsize=18 crowd=2
               projection=aitoff viewsys=galactic
               layer1=mark size1=0
               shading1=density densemap1=gnuplot2 densefunc1=log
               densesub1=0.5,.95 denseclip1=0.02,1
               in1=gums_mw_all.fits
               lon1=alpha lat1=delta datasys1=equatorial
```

```
icmd1=progress out=mw.pdf
```

Makes an all-sky plot using a Hammer-Aitoff projection into galactic coordinates of a large dataset. Density shading means that the colour at each point is dependent on how many points are plotted; the density colour map has been fine-tuned here to get a specific visual effect. The sky coordinates in the input file (alpha and delta) are equatorial, but these are transformed to galactic coordinates for plotting. The progress filter applied to the input table displays a progress indicator on the console to see how far it's got. The result is written to a PDF file.

This command was used to plot the GUMS-10 MW dataset, a simulation of the milky way stars seen by the Gaia satellite; The 2.1 billion row plot took about 45 minutes.

B.12 `plot2cube`: Draws a cube plot

`plot2cube` draws plots in a Cartesian 3-dimensional space. The plotting volume is a cube, which is viewed from the outside and usually bounded by an annotated wire frame.

Positional coordinates are specified as `x`, `y`, `z` triples, e.g.:

```
plot2cube layer1=mark in1=sim.fits x1=XPOS y1=YPOS z1=ZPOS
```

Content is added to the plot by specifying one or more *plot layers* using the `layerN` parameter. The `N` part is a suffix applied to all the parameters affecting a given layer; any suffix (including the empty string) may be used. Available layers for this plot type are: `mark` (Section 8.3.1), `size` (Section 8.3.2), `sizexy` (Section 8.3.3), `xyzvector` (Section 8.3.42), `xyzerror` (Section 8.3.43), `link2` (Section 8.3.8), `mark2` (Section 8.3.9), `poly4` (Section 8.3.10), `mark4` (Section 8.3.11), `polygon` (Section 8.3.12), `label` (Section 8.3.24), `line3d` (Section 8.3.44), `contour` (Section 8.3.26), `spheregrid` (Section 8.3.45).

B.12.1 Usage

The usage of `plot2cube` is

```
stilts <stilts-flags> plot2cube xpix=<int-value> ypix=<int-value>
                                insets=<top>,<left>,<bottom>,<right>
                                omode=swing|out|cgi|discard|auto
                                storage=simple|memory|disk|policy|cache|basic-cache|persist
                                seq=<suffix>[,...] legend=true|false
                                legborder=true|false legopaque=true|false
                                legseq=<suffix>[,...] legpos=<xfrac,yfrac>
                                title=<value>
                                auxmap=<map-name>|<color>-<color>[-<color>...]
                                auxclip=<lo>,<hi> auxflip=true|false
                                auxquant=<number>
                                auxfunc=log|linear|histogram|histolog|sqrt|square|acos|cos
                                auxmin=<number> auxmax=<number>
                                auxlabel=<text> auxcrowd=<factor>
                                auxwidth=<pixels> auxvisible=true|false
                                forcebitmap=true|false compositor=0..1
                                animate=<table> afmt=<in-format>
                                astream=true|false acmd=<cmds>
                                parallel=<int-value> xlog=true|false
                                ylog=true|false zlog=true|false
                                xflip=true|false yflip=true|false
                                zflip=true|false xlabel=<text>
                                ylabel=<text> zlabel=<text> xcrowd=<number>
                                ycrowd=<number> zcrowd=<number>
                                frame=true|false minor=true|false
                                gridaa=true|false
                                texttype=plain|antialias|latex
                                fontsize=<int-value>
                                fontstyle=standard|serif|mono
                                fontweight=plain|bold|italic|bold_italic
```

```
xmin=<number> xmax=<number> xsub=<lo>,<hi>
ymin=<number> ymax=<number> ysub=<lo>,<hi>
zmin=<number> zmax=<number> zsub=<lo>,<hi>
phi=<degrees> theta=<degrees> psi=<degrees>
zoom=<factor> xoff=<pixels> yoff=<pixels>
zoomaxes=[[x][y][z]] zoomfactor=<number>
leglabelN=<text>
layerN=<layer-type> <layerN-specific-params>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.plot2.task.CubePlot2Task`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`acmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the animation control table as specified by parameter `animate`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`afmt = <in-format>` ***(String)***

Specifies the format of the animation control table as specified by parameter `animate`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`animate = <table>` ***(StarTable)***

If not null, this parameter causes the command to create a sequence of plots instead of just one. The parameter value is a table with one row for each frame to be produced. Columns in the table are interpreted as parameters which may take different values for each frame; the column name is the parameter name, and the value for a given frame is its value from that row. Animating like this is considerably more efficient than invoking the STILTS command in a loop.

The location of the animation control table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `afmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`astream = true|false` ***(Boolean)***

If set true, the animation control table specified by the `animate` parameter will be read as a stream. It is necessary to give the `afmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`auxclip = <lo>,<hi>` ***(Subrange)***

Defines a subrange of the colour ramp to be used for Aux shading. The value is specified as a (low,high) comma-separated pair of two numbers between 0 and 1.

If the full range `0,1` is used, the whole range of colours specified by the selected shader will be used. But if for instance a value of `0,0.5` is given, only those colours at the left hand end of the ramp will be seen.

If the null (default) value is chosen, a default clip will be used. This generally covers most or all of the range 0-1 but for colour maps which fade to white, a small proportion of the lower end may be excluded, to ensure that all the colours are visually distinguishable from a white background. This default is usually a good idea if the colour map is being used with something like a scatter plot, where markers are plotted against a white background. However, for something like a density map when the whole plotting area is tiled with colours from the map, it may be better to supply the whole range `0,1` explicitly.

`auxcrowd = <factor>` ***(Double)***

Determines how closely the tick marks are spaced on the Aux axis, if visible. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

[Default: `1.0`]

`auxflip = true|false` ***(Boolean)***

If true, the colour map on the Aux axis will be reversed.

[Default: `false`]

`auxfunc = log|linear|histogram|histolog|sqrt|square|acos|cos` ***(Scaling)***

Defines the way that values in the Aux range are mapped to the selected colour ramp.

The available options are:

- `log`: Logarithmic scaling
- `linear`: Linear scaling
- `histogram`: Scaling follows data distribution, with linear axis
- `histolog`: Scaling follows data distribution, with logarithmic axis
- `sqrt`: Square root scaling
- `square`: Square scaling
- `acos`: Arccos Scaling
- `cos`: Cos Scaling

For all these options, the full range of data values is used, and displayed on the colour bar if applicable. The `Linear`, `Log`, `Square` and `Sqrt` options just apply the named function to the full data range. The histogram options on the other hand use a scaling function that corresponds to the actual distribution of the data, so that there are about the same number of points (or pixels, or whatever is being scaled) of each colour. The histogram options are somewhat more expensive, but can be a good choice if you are exploring data whose distribution is unknown or not well-behaved over its min-max range. The `Histogram` and `HistoLog` options both assign

the colours in the same way, but they display the colour ramp with linear or logarithmic annotation respectively; the `HistoLog` option also ignores non-positive values.

[Default: `linear`]

`auxlabel = <text>` ***(String)***
Sets the label used to annotate the aux axis, if it is visible.

`auxmap = <map-name>|<color>-<color>[-<color>...]` ***(Shader)***
Color map used for Aux axis shading.

A mixed bag of colour ramps are available: `inferno`, `magma`, `plasma`, `viridis`, `cividis`, `cubehelix`, `sron`, `rainbow`, `rainbow2`, `rainbow3`, `pastel`, `accent`, `gnuplot`, `gnuplot2`, `specxby`, `set1`, `paired`, `hotcold`, `rdbu`, `piyg`, `brbg`, `cyan-magenta`, `red-blue`, `brg`, `heat`, `cold`, `light`, `greyscale`, `colour`, `standard`, `bugn`, `bupu`, `orrd`, `pubu`, `purd`, `painbow`, `huecl`, `hue`, `intensity`, `rgb_red`, `rgb_green`, `rgb_blue`, `hsv_h`, `hsv_s`, `hsv_v`, `yuv_y`, `yuv_u`, `yuv_v`, `scale_hsv_s`, `scale_hsv_v`, `scale_yuv_y`, `mask`, `blacker`, `whiter`, `transparency`. *Note:* many of these, including rainbow-like ones, are frowned upon by the visualisation community.

You can also construct your own custom colour map by giving a sequence of colour names separated by minus sign ("`-`") characters. In this case the ramp is a linear interpolation between each pair of colours named, using the same syntax as when specifying a colour value. So for instance "`yellow-hotpink-#0000ff`" would shade from yellow via hot pink to blue.

[Default: `inferno`]

`auxmax = <number>` ***(Double)***
Maximum value of the data coordinate on the Aux axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`auxmin = <number>` ***(Double)***
Minimum value of the data coordinate on the Aux axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`auxquant = <number>` ***(Double)***
Allows the colour map used for the Aux axis to be quantised. If an integer value N is chosen then the colour map will be viewed as N discrete evenly-spaced levels, so that only N different colours will appear in the plot. This can be used to generate a contour-like effect, and may make it easier to trace the boundaries of regions of interest by eye.

If left blank, the colour map is nominally continuous (though in practice it may be quantised to a medium-sized number like 256).

`auxvisible = true|false` ***(Boolean)***
Determines whether the aux axis colour ramp is displayed alongside the plot.

If not supplied (the default), the aux axis will be visible when aux shading is used in any of the plotted layers.

`auxwidth = <pixels>` ***(Integer)***
Determines the lateral size of the aux colour ramp, if visible, in pixels.

[Default: `15`]

`compositor = 0..1` ***(Compositor)***
Defines how multiple overplotted partially transparent pixels are combined to form a resulting colour. The way this is used depends on the details of the specified plot.

Currently, this parameter takes a "boost" value in the range 0..1. If the value is zero, saturation semantics are used: RGB colours are added in proportion to their associated alpha value until the total alpha is saturated (reaches 1), after which additional pixels have no further effect. For larger boost values, the effect is similar, but any non-zero alpha in the output is boosted to the given minimum value. The effect of this is that even very slightly populated pixels can be visually distinguished from unpopulated ones which may not be the case for saturation

composition.

[Default: `0.05`]

`fontsize = <int-value>` ***(Integer)***
Size of the text font in points.

[Default: `12`]

`fontstyle = standard|serif|mono` ***(FontType)***
Font style for text.

The available options are:

- `standard`
- `serif`
- `mono`

[Default: `standard`]

`fontweight = plain|bold|italic|bold_italic` ***(FontWeight)***
Font weight for text.

The available options are:

- `plain`
- `bold`
- `italic`
- `bold_italic`

[Default: `plain`]

`forcebitmap = true|false` ***(Boolean)***
Affects whether rendering of the data contents of a plot (though not axis labels etc) is always done to an intermediate bitmap rather than, where possible, being painted using graphics primitives. This is a rather arcane setting that may nevertheless have noticeable effects on the appearance and size of an output graphics file, as well as plotting time. For some types of plot (e.g. `shadingN=auto` or `shadingN=density`) it will have no effect, since this kind of rendering happens in any case.

When writing to vector graphics formats (PDF and PostScript), setting it true will force the data contents to be bitmapped. This may make the output less beautiful (round markers will no longer be perfectly round), but it may result in a much smaller file if there are very many data points.

When writing to bitmapped output formats (PNG, GIF, JPEG, ...), it fixes shapes to be the same as seen on the screen rather than be rendered at the mercy of the graphics system, which sometimes introduces small distortions.

[Default: `false`]

`frame = true|false` ***(Boolean)***
If true, a cube wire frame with labelled axes is drawn to indicate the limits of the plotted 3D region. If false, no wire frame and no axes are drawn.

[Default: `true`]

`gridaa = true|false` ***(Boolean)***
If true, grid lines are drawn with antialiasing. Antialiased lines look smoother, but may take perceptibly longer to draw. Only has any effect for bitmapped output formats.

[Default: `false`]

`insets = <top>,<left>,<bottom>,<right>` ***(Padding)***
Defines the amount of space in pixels around the actual plotting area. This space is used for

axis labels, and other decorations and any left over forms an empty border.

The size and position of the actual plotting area is determined by this parameter along with `xpix` and `ypix`.

The value of this parameter is 4 comma separated integers: `<top>,<left>,<bottom>,<right>`. Any or all of these values may be left blank, in which case the corresponding margin will be calculated automatically according to how much space is required.

`layerN = <layer-type> <layerN-specific-params>` ***(LayerType)***

Selects one of the available plot types for layerN. A plot consists of a plotting surface, set up using the various unsuffixed parameters of the plotting command, and zero or more plot layers. Each layer is introduced by a parameter with the name `layer<N>` where the suffix "`<N>`" is a label identifying the layer and is appended to all the parameter names which configure that layer. Suffixes may be any string, including the empty string.

This parameter may take one of the following values, described in more detail in Section 8.3:

- `mark (Section 8.3.1)`
- `size (Section 8.3.2)`
- `sizexy (Section 8.3.3)`
- `xyzvector (Section 8.3.42)`
- `xyzerror (Section 8.3.43)`
- `link2 (Section 8.3.8)`
- `mark2 (Section 8.3.9)`
- `poly4 (Section 8.3.10)`
- `mark4 (Section 8.3.11)`
- `polygon (Section 8.3.12)`
- `label (Section 8.3.24)`
- `line3d (Section 8.3.44)`
- `contour (Section 8.3.26)`
- `spheregrid (Section 8.3.45)`

Each of these layer types comes with a list of type-specific parameters to define the details of that layer, including some or all of the following groups:

- input table parameters (e.g. `inN`, `icmdN`)
- coordinate params referring to input table columns (e.g. `xN`, `yN`)
- layer style parameters (e.g. `shadingN`, `colorN`)

Every parameter notionally carries the same suffix `N`. However, if the suffix is not present, the application will try looking for a parameter with the same name with no suffix instead. In this way, if several layers have the same value for a given parameter (for instance input table), you can supply it using one unsuffixed parameter to save having to supply several parameters with the same value but different suffixes.

`legborder = true|false` ***(Boolean)***

If true, a line border is drawn around the legend.

[Default: `true`]

`legend = true|false` ***(Boolean)***

Whether to draw a legend or not. If no value is supplied, the decision is made automatically: a legend is drawn only if it would have more than one entry.

`leglabelN = <text>` ***(String)***

Sets the presentation label for the layer with a given suffix. This is the text which is displayed in the legend, if present. Multiple layers may use the same label, in which case they will be combined to form a single legend entry.

If no value is supplied (the default), the suffix itself is used as the label.

`legopaque = true|false` ***(Boolean)***
If true, the background of the legend is opaque, and the legend obscures any plot components behind it. Otherwise, it's transparent.

[Default: `true`]

`legpos = <xfrac,yfrac>` ***(double[])***
Determines the internal position of the legend on the plot. The value is a comma-separated pair of values giving the X and Y positions of the legend within the plotting bounds, so for instance "`0.5,0.5`" will put the legend right in the middle of the plot. If no value is supplied, the legend will appear outside the plot boundary.

`legseq = <suffix>[,...]` ***(String[])***
Determines which layers are represented in the legend (if present) and in which order they appear. The legend has a line for each layer label (as determined by the `leglabelN` parameter). If multiple layers have the same label, they will contribute to the same entry in the legend, with style icons plotted over each other. The value of this parameter is a comma-separated sequence of layer suffixes, which determines the order in which the legend entries appear. Layers with suffixes missing from this list do not show up in the legend at all.

If no value is supplied (the default), the sequence is the same as the layer plotting sequence (see `seq`).

`minor = true|false` ***(Boolean)***
If true, minor tick marks are painted along the axes as well as the major tick marks. Minor tick marks do not have associated grid lines.

[Default: `true`]

`omode = swing|out|cgi|discard|auto` ***(PaintMode)***
Determines how the drawn plot will be output, see Section 8.5.

- `swing`: Plot will be displayed in a window on the screen. This plot is "live"; it can be resized and (except for old-style plots) navigated around with mouse actions in the same way as plots in TOPCAT.
- `out`: Plot will be written to a file given by `out` using the graphics format given by `ofmt`.
- `cgi`: Plot will be written in a way suitable for CGI use direct from a web server. The output is in the graphics format given by `ofmt`, preceded by a suitable "Content-type" declaration.
- `discard`: Plot is drawn, but discarded. There is no output.
- `auto`: Behaves as `swing` or `out` mode depending on presence of `out` parameter

[Default: `auto`]

`parallel = <int-value>` ***(Integer)***
Determines how many threads will run in parallel if animation output is being produced. Only used if the `animate` parameter is supplied. The default value is the number of processors apparently available to the JVM.

[Default: `20`]

`phi = <degrees>` ***(Double)***
First of the Euler angles, in the ZXZ sequence, defining the rotation of the plotted 3d space. Units are degrees. This is the rotation around the initial Z axis applied before the plot is viewed.

[Default: `30`]

`psi = <degrees>` ***(Double)***
Second of the Euler angles, in the ZXZ sequence, defining the rotation of the plotted 3d space. Units are degrees.

[Default: `0`]

`seq = <suffix>[,...]` ***(String[])***

Contains a comma-separated list of layer suffixes to determine the order in which layers are drawn on the plot. This can affect which symbol are plotted on top of, and so potentially obscure, which other ones.

When specifying a plot, multiple layers may be specified, each introduced by a parameter `layer<N>`, where `<N>` is a different (arbitrary) suffix labelling the layer, and is appended to all the parameters specific to defining that layer.

By default the layers are drawn on the plot in the order in which the `layer*` parameters appear on the command line. However if this parameter is specified, each comma-separated element is interpreted as a layer suffix, giving the ordered list of layers to plot. Every element of the list must be a suffix with a corresponding `layer` parameter, but missing or repeated elements are allowed.

`storage = simple|memory|disk|policy|cache|basic-cache|persistent|parallel`
(DataStoreFactory)

Determines the way that data is accessed when constructing the plot. There are two main options, cached or not. If no caching is used then rows are read sequentially from the specified input table(s) every time they are required. This generally requires a small resource footprint (though that can depend on how the table is specified) and makes sense if the data only needs to be scanned once or perhaps if the table is very large. If caching is used then the required data is read once from the specified input table(s), then prepared and cached before any plotting is performed, and plots are done using this cached data. This may use a significant amount of storage for large tables but it's usually more sensible (faster) if the data will need to be scanned multiple times. There are various options for cache storage.

The options are:

- `simple`: no caching, data read directly from input table
- `memory`: cached to memory; OutOfMemoryError possible for very large plots
- `disk`: cached to disk
- `policy`: cached using application-wide default storage policy, which is usually *adaptive* (memory/disk hybrid)
- `persistent`: cached to persistent files on disk, in the system temporary directory (defined by system property `java.io.tmpdir`). If this is used, plot data will be stored on disk in a way that means they can be re-used between STILTS invocations, so data preparation can be avoided on subsequent runs. Note however it can leave potentially large files in your temporary directory.
- `cache`: synonym for `memory` (backward compatibility)
- `basic-cache`: dumber version of `memory` (no optimisation for constant-valued columns)
- `parallel`: experimental version of memory-based cache that reads into the cache in parallel for large files. This will make the plot faster to prepare, but interaction is a bit slower and sequence-dependent attributes of the plot may not come out right. This experimental option may be withdrawn or modified in future releases.

The default value is `memory` if a live plot is being generated (`omode=swing`), since in that case the plot needs to be redrawn every time the user performs plot navigation actions or resizes the window, or if animations are being produced. Otherwise (e.g. output to a graphics file) the default is `simple`.

[Default: `simple`]

`texttype = plain|antialias|latex` ***(TextSyntax)***

Determines how to turn label text into characters on the plot. `Plain` and `Antialias` both take the text at face value, but `Antialias` smooths the characters. `LaTeX` interprets the text as LaTeX source code and typesets it accordingly.

When not using LaTeX, antialiased text usually looks nicer, but can be perceptibly slower to plot. At time of writing, on MacOS antialiased text seems to be required to stop the writing

coming out upside-down for non-horizontal text (MacOS java bug).

[Default: `plain`]

`theta = <degrees>` ***(Double)***
Second of the Euler angles, in the ZXZ sequence, defining the rotation of the plotted 3d space. Units are degrees. This is the rotation towards the viewer.

[Default: `-15`]

`title = <value>` ***(String)***
Text of a title to be displayed at the top of the plot. If null, the default, no title is shown and there's more space for the graphics.

`xcrowd = <number>` ***(Double)***
Determines how closely the tick marks are spaced on the X axis. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

[Default: `1`]

`xflip = true|false` ***(Boolean)***
If true, the scale on the X axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

`xlabel = <text>` ***(String)***
Gives a label to be used for annotating axis X A default value based on the plotted data will be used if no value is supplied.

[Default: `X`]

`xlog = true|false` ***(Boolean)***
If false (the default), the scale on the X axis is linear, if true it is logarithmic.

[Default: `false`]

`xmax = <number>` ***(Double)***
Maximum value of the data coordinate on the X axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`xmin = <number>` ***(Double)***
Minimum value of the data coordinate on the X axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`xoff = <pixels>` ***(Double)***
Shifts the whole plot within the plotting region by the given number of pixels in the horizontal direction.

[Default: `0`]

`xpix = <int-value>` ***(Integer)***
Size of the output image in the X direction in pixels. This includes space for any axis labels, padding and other decoration outside the plot area itself. See also `insets`.

[Default: `500`]

`xsub = <lo>,<hi>` ***(Subrange)***
Defines a normalised adjustment to the data range of the X axis. The value may be specified as a comma-separated pair of two numbers, giving the lower and upper bounds of the range of of interest respectively. This sub-range is applied to the data range that would otherwise be used, either automatically calculated or explicitly supplied; zero corresponds to the lower bound and one to the upper.

The default value "0,1" therefore has no effect. The range could be restricted to its lower half with the value 0,0.5.

[Default: 0,1]

ycrowd = <number> ***(Double)***
Determines how closely the tick marks are spaced on the Y axis. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

[Default: 1]

yflip = true|false ***(Boolean)***
If true, the scale on the Y axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: false]

ylabel = <text> ***(String)***
Gives a label to be used for annotating axis Y A default value based on the plotted data will be used if no value is supplied.

[Default: Y]

ylog = true|false ***(Boolean)***
If false (the default), the scale on the Y axis is linear, if true it is logarithmic.

[Default: false]

ymax = <number> ***(Double)***
Maximum value of the data coordinate on the Y axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

ymin = <number> ***(Double)***
Minimum value of the data coordinate on the Y axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

yoff = <pixels> ***(Double)***
Shifts the whole plot within the plotting region by the given number of pixels in the vertical direction.

[Default: 0]

ypix = <int-value> ***(Integer)***
Size of the output image in the Y direction in pixels. This includes space for any axis labels, padding and other decoration outside the plot area itself. See also insets.

[Default: 400]

ysub = <lo>,<hi> ***(Subrange)***
Defines a normalised adjustment to the data range of the Y axis. The value may be specified as a comma-separated pair of two numbers, giving the lower and upper bounds of the range of of interest respectively. This sub-range is applied to the data range that would otherwise be used, either automatically calculated or explicitly supplied; zero corresponds to the lower bound and one to the upper.

The default value "0,1" therefore has no effect. The range could be restricted to its lower half with the value 0,0.5.

[Default: 0,1]

zcrowd = <number> ***(Double)***
Determines how closely the tick marks are spaced on the Z axis. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get

very closely spaced marks you may need to reduce the font size as well.

[Default: 1]

zflip = true|false ***(Boolean)***
If true, the scale on the Z axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: false]

zlabel = <text> ***(String)***
Gives a label to be used for annotating axis Z A default value based on the plotted data will be used if no value is supplied.

[Default: Z]

zlog = true|false ***(Boolean)***
If false (the default), the scale on the Z axis is linear, if true it is logarithmic.

[Default: false]

zmax = <number> ***(Double)***
Maximum value of the data coordinate on the Z axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

zmin = <number> ***(Double)***
Minimum value of the data coordinate on the Z axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

zoom = <factor> ***(Double)***
Sets the magnification factor at which the the plotted 3D region itself is viewed, without affecting its contents. The default value is 1, which means the cube fits into the plotting space however it is rotated. Much higher zoom factors will result in parts of the plotting region and axes being drawn outside of the plotting region (so invisible).

[Default: 1]

zoomaxes = [[x][y][z]] ***(boolean[])***
Determines which axes are affected by zoom navigation actions.

If no value is supplied (the default), the mouse wheel zooms around the center of the cube, and right-button (or CTRL-) drag zooms in the two dimensions most closely aligned with the plane of the screen, with the reference position set by the initial position of the mouse.

If this value is set (legal values are x, y, z, xy, yz, xz and xyz) then all zoom operations are around the cube center and affect the axes named.

zoomfactor = <number> ***(Double)***
Sets the amount by which the plot view zooms in or out for each unit of mouse wheel movement. A value of 1 means that mouse wheel zooming has no effect. A higher value means that the mouse wheel zooms faster and a value nearer 1 means it zooms slower. Values below 1 are not permitted.

[Default: 1.2]

zsub = <lo>,<hi> ***(Subrange)***
Defines a normalised adjustment to the data range of the Z axis. The value may be specified as a comma-separated pair of two numbers, giving the lower and upper bounds of the range of of interest respectively. This sub-range is applied to the data range that would otherwise be used, either automatically calculated or explicitly supplied; zero corresponds to the lower bound and one to the upper.

The default value "0,1" therefore has no effect. The range could be restricted to its lower half with the value 0,0.5.

[Default: 0,1]

B.12.2 Examples

Some examples of `plot2cube` are shown below. See Section 8.1.3 for some examples of producing **animations**, for instance of a rotating cube.

```
stilts plot2cube
```

Just displays a unit cube wireframe in a window. You can rotate it with the mouse.

```
stilts plot2cube layer.1=mark in.1=sim.fits x.1=x y.1=y z.1=z
                 shading.1=density densemap.1=pastel
```

Plots markers with x,y,z positions on the screen. You can rotate, zoom and pan the cube on the window this produces. Density shading is used, which means you can see the lines of sight along which most objects fall, though single points are still visible. Density shading is usually a good choice if there is just one dataset, though it can get confusing with more than one.

```
stilts plot2cube in=gavo_g2.fits
                 x=X y=Y z=Z
                 shading=aux aux=HALOID opaque=2.5 auxmap=red-blue
                 layer_m=mark shape_m=open_circle size_m=2
                 layer_v=xyzvector xdelta_v=velX ydelta_v=velY zdelta_v=velZ
```

Plots points in three dimensions with little arrows representing velocity as well as position markers; layer `_m` draws the markers and layer `_v` draws the arrows. Points and vectors are coloured according to the HALOID data value. The positional coordinates (x, y, z) and the shading options are common to both layers, so they can be specified without a prefix.

B.13 `plot2sphere`: Draws a sphere plot

`plot2sphere` draws plots in an isotropic 3-dimensional space using spherical polar coordinates. The plotting volume is a cube, which is viewed from the outside and usually bounded by a wire frame annotated by Cartesian coordinates. This viewing cube is not necessarily centered on the coordinate origin.

This plotting geometry is like that used by `plot2cube`, but the coordinate unit size is always the same in the three dimensions, and the coordinates are specified differently.

Positional coordinates are specified as `lon`, `lat`, `r` triples, e.g.:

```
plot2sphere layer1=mark in1=survey.fits lon1=RA lat1=DEC r1=REDSHIFT
```

Content is added to the plot by specifying one or more *plot layers* using the `layerN` parameter. The `N` part is a suffix applied to all the parameters affecting a given layer; any suffix (including the empty string) may be used. Available layers for this plot type are: `mark` (Section 8.3.1), `size` (Section 8.3.2), `sizexy` (Section 8.3.3), `link2` (Section 8.3.8), `mark2` (Section 8.3.9), `poly4` (Section 8.3.10), `mark4` (Section 8.3.11), `polygon` (Section 8.3.12), `area` (Section 8.3.13), `central` (Section 8.3.14), `label` (Section 8.3.24), `arealabel` (Section 8.3.25), `line3d` (Section 8.3.44), `contour` (Section 8.3.26), `spheregrid` (Section 8.3.45).

B.13.1 Usage

The usage of `plot2sphere` is

```
stilts <stilts-flags> plot2sphere xpix=<int-value> ypix=<int-value>
                                  insets=<top>,<left>,<bottom>,<right>
                                  omode=swing|out|cgi|discard|auto
                                  storage=simple|memory|disk|policy|cache|basic-cache|persi
                                  seq=<suffix>[,...] legend=true|false
                                  legborder=true|false legopaque=true|false
                                  legseq=<suffix>[,...]
                                  legpos=<xfrac,yfrac> title=<value>
                                  auxmap=<map-name>|<color>-<color>[-<color>...]
                                  auxclip=<lo>,<hi> auxflip=true|false
                                  auxquant=<number>
                                  auxfunc=log|linear|histogram|histolog|sqrt|square|acos|co
                                  auxmin=<number> auxmax=<number>
                                  auxlabel=<text> auxcrowd=<factor>
                                  auxwidth=<pixels> auxvisible=true|false
                                  forcebitmap=true|false compositor=0..1
                                  animate=<table> afmt=<in-format>
                                  astream=true|false acmd=<cmds>
                                  parallel=<int-value> crowd=<number>
                                  frame=true|false minor=true|false
                                  gridaa=true|false
                                  texttype=plain|antialias|latex
                                  fontsize=<int-value>
                                  fontstyle=standard|serif|mono
                                  fontweight=plain|bold|italic|bold_italic
                                  cx=<number> cy=<number> cz=<number>
                                  scale=<number> phi=<degrees>
                                  theta=<degrees> psi=<degrees>
                                  zoom=<factor> xoff=<pixels> yoff=<pixels>
                                  zoomfactor=<number> leglabelN=<text>
                                  layerN=<layer-type> <layerN-specific-params>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.plot2.task.SpherePlot2Task`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`acmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the animation control table as specified by parameter `animate`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`afmt = <in-format>` ***(String)***

Specifies the format of the animation control table as specified by parameter `animate`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`animate = <table>` ***(StarTable)***

If not null, this parameter causes the command to create a sequence of plots instead of just one. The parameter value is a table with one row for each frame to be produced. Columns in the

table are interpreted as parameters which may take different values for each frame; the column name is the parameter name, and the value for a given frame is its value from that row. Animating like this is considerably more efficient than invoking the STILTS command in a loop.

The location of the animation control table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `afmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`astream = true|false` ***(Boolean)***

If set true, the animation control table specified by the `animate` parameter will be read as a stream. It is necessary to give the `afmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`auxclip = <lo>,<hi>` ***(Subrange)***

Defines a subrange of the colour ramp to be used for Aux shading. The value is specified as a (low,high) comma-separated pair of two numbers between 0 and 1.

If the full range `0,1` is used, the whole range of colours specified by the selected shader will be used. But if for instance a value of `0,0.5` is given, only those colours at the left hand end of the ramp will be seen.

If the null (default) value is chosen, a default clip will be used. This generally covers most or all of the range 0-1 but for colour maps which fade to white, a small proportion of the lower end may be excluded, to ensure that all the colours are visually distinguishable from a white background. This default is usually a good idea if the colour map is being used with something like a scatter plot, where markers are plotted against a white background. However, for something like a density map when the whole plotting area is tiled with colours from the map, it may be better to supply the whole range `0,1` explicitly.

`auxcrowd = <factor>` ***(Double)***

Determines how closely the tick marks are spaced on the Aux axis, if visible. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

[Default: `1.0`]

`auxflip = true|false` ***(Boolean)***

If true, the colour map on the Aux axis will be reversed.

[Default: `false`]

`auxfunc = log|linear|histogram|histolog|sqrt|square|acos|cos` ***(Scaling)***

Defines the way that values in the Aux range are mapped to the selected colour ramp.

The available options are:

- `log`: Logarithmic scaling
- `linear`: Linear scaling
- `histogram`: Scaling follows data distribution, with linear axis
- `histolog`: Scaling follows data distribution, with logarithmic axis
- `sqrt`: Square root scaling
- `square`: Square scaling
- `acos`: Arccos Scaling
- `cos`: Cos Scaling

For all these options, the full range of data values is used, and displayed on the colour bar if applicable. The `Linear`, `Log`, `Square` and `Sqrt` options just apply the named function to the full data range. The histogram options on the other hand use a scaling function that corresponds to the actual distribution of the data, so that there are about the same number of points (or pixels, or whatever is being scaled) of each colour. The histogram options are somewhat more expensive, but can be a good choice if you are exploring data whose distribution is unknown or not well-behaved over its min-max range. The `Histogram` and `HistoLog` options both assign the colours in the same way, but they display the colour ramp with linear or logarithmic annotation respectively; the `HistoLog` option also ignores non-positive values.

[Default: `linear`]

`auxlabel = <text>` ***(String)***
Sets the label used to annotate the aux axis, if it is visible.

`auxmap = <map-name>|<color>-<color>[-<color>...]` ***(Shader)***
Color map used for Aux axis shading.

A mixed bag of colour ramps are available: `inferno`, `magma`, `plasma`, `viridis`, `cividis`, `cubehelix`, `sron`, `rainbow`, `rainbow2`, `rainbow3`, `pastel`, `accent`, `gnuplot`, `gnuplot2`, `specxby`, `set1`, `paired`, `hotcold`, `rdbu`, `piyg`, `brbg`, `cyan-magenta`, `red-blue`, `brg`, `heat`, `cold`, `light`, `greyscale`, `colour`, `standard`, `bugn`, `bupu`, `orrd`, `pubu`, `purd`, `painbow`, `huecl`, `hue`, `intensity`, `rgb_red`, `rgb_green`, `rgb_blue`, `hsv_h`, `hsv_s`, `hsv_v`, `yuv_y`, `yuv_u`, `yuv_v`, `scale_hsv_s`, `scale_hsv_v`, `scale_yuv_y`, `mask`, `blacker`, `whiter`, `transparency`. *Note:* many of these, including rainbow-like ones, are frowned upon by the visualisation community.

You can also construct your own custom colour map by giving a sequence of colour names separated by minus sign ("`-`") characters. In this case the ramp is a linear interpolation between each pair of colours named, using the same syntax as when specifying a colour value. So for instance "`yellow-hotpink-#0000ff`" would shade from yellow via hot pink to blue.

[Default: `inferno`]

`auxmax = <number>` ***(Double)***
Maximum value of the data coordinate on the Aux axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`auxmin = <number>` ***(Double)***
Minimum value of the data coordinate on the Aux axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

`auxquant = <number>` ***(Double)***
Allows the colour map used for the Aux axis to be quantised. If an integer value N is chosen then the colour map will be viewed as N discrete evenly-spaced levels, so that only N different colours will appear in the plot. This can be used to generate a contour-like effect, and may make it easier to trace the boundaries of regions of interest by eye.

If left blank, the colour map is nominally continuous (though in practice it may be quantised to a medium-sized number like 256).

`auxvisible = true|false` ***(Boolean)***
Determines whether the aux axis colour ramp is displayed alongside the plot.

If not supplied (the default), the aux axis will be visible when aux shading is used in any of the plotted layers.

auxwidth = <pixels> ***(Integer)***
Determines the lateral size of the aux colour ramp, if visible, in pixels.

[Default: 15]

compositor = 0..1 ***(Compositor)***
Defines how multiple overplotted partially transparent pixels are combined to form a resulting colour. The way this is used depends on the details of the specified plot.

Currently, this parameter takes a "boost" value in the range 0..1. If the value is zero, saturation semantics are used: RGB colours are added in proporition to their associated alpha value until the total alpha is saturated (reaches 1), after which additional pixels have no further effect. For larger boost values, the effect is similar, but any non-zero alpha in the output is boosted to the given minimum value. The effect of this is that even very slightly populated pixels can be visually distinguished from unpopulated ones which may not be the case for saturation composition.

[Default: 0.05]

crowd = <number> ***(Double)***
Determines how closely tick marks are spaced on the wire frame axes. The default value is 1, meaning normal crowding. Larger values result in more grid lines, and smaller values in fewer grid lines.

[Default: 1]

cx = <number> ***(Double)***
Gives the central coordinate in the X dimension. This will be determined from the data range if not supplied.

cy = <number> ***(Double)***
Gives the central coordinate in the Y dimension. This will be determined from the data range if not supplied.

cz = <number> ***(Double)***
Gives the central coordinate in the Z dimension. This will be determined from the data range if not supplied.

fontsize = <int-value> ***(Integer)***
Size of the text font in points.

[Default: 12]

fontstyle = standard|serif|mono ***(FontType)***
Font style for text.

The available options are:

- standard
- serif
- mono

[Default: standard]

fontweight = plain|bold|italic|bold_italic ***(FontWeight)***
Font weight for text.

The available options are:

- plain
- bold
- italic

- bold_italic

[Default: plain]

forcebitmap = true|false ***(Boolean)***

Affects whether rendering of the data contents of a plot (though not axis labels etc) is always done to an intermediate bitmap rather than, where possible, being painted using graphics primitives. This is a rather arcane setting that may nevertheless have noticeable effects on the appearance and size of an output graphics file, as well as plotting time. For some types of plot (e.g. shadingN=auto or shadingN=density) it will have no effect, since this kind of rendering happens in any case.

When writing to vector graphics formats (PDF and PostScript), setting it true will force the data contents to be bitmapped. This may make the output less beautiful (round markers will no longer be perfectly round), but it may result in a much smaller file if there are very many data points.

When writing to bitmapped output formats (PNG, GIF, JPEG, ...), it fixes shapes to be the same as seen on the screen rather than be rendered at the mercy of the graphics system, which sometimes introduces small distortions.

[Default: false]

frame = true|false ***(Boolean)***

If true, a cube wire frame with labelled axes is drawn to indicate the limits of the plotted 3D region. If false, no wire frame and no axes are drawn.

[Default: true]

gridaa = true|false ***(Boolean)***

If true, grid lines are drawn with antialiasing. Antialiased lines look smoother, but may take perceptibly longer to draw. Only has any effect for bitmapped output formats.

[Default: false]

insets = <top>,<left>,<bottom>,<right> ***(Padding)***

Defines the amount of space in pixels around the actual plotting area. This space is used for axis labels, and other decorations and any left over forms an empty border.

The size and position of the actual plotting area is determined by this parameter along with xpix and ypix.

The value of this parameter is 4 comma separated integers: <top>,<left>,<bottom>,<right>. Any or all of these values may be left blank, in which case the corresponding margin will be calculated automatically according to how much space is required.

layerN = <layer-type> <layerN-specific-params> ***(LayerType)***

Selects one of the available plot types for layerN. A plot consists of a plotting surface, set up using the various unsuffixed parameters of the plotting command, and zero or more plot layers. Each layer is introduced by a parameter with the name layer<N> where the suffix "<N>" is a label identifying the layer and is appended to all the parameter names which configure that layer. Suffixes may be any string, including the empty string.

This parameter may take one of the following values, described in more detail in Section 8.3:

- mark (Section 8.3.1)
- size (Section 8.3.2)
- sizexy (Section 8.3.3)
- link2 (Section 8.3.8)
- mark2 (Section 8.3.9)
- poly4 (Section 8.3.10)
- mark4 (Section 8.3.11)
- polygon (Section 8.3.12)

- area (Section 8.3.13)
- central (Section 8.3.14)
- label (Section 8.3.24)
- arealabel (Section 8.3.25)
- line3d (Section 8.3.44)
- contour (Section 8.3.26)
- spheregrid (Section 8.3.45)

Each of these layer types comes with a list of type-specific parameters to define the details of that layer, including some or all of the following groups:

- input table parameters (e.g. inN, icmdN)
- coordinate params referring to input table columns (e.g. xN, yN)
- layer style parameters (e.g. shadingN, colorN)

Every parameter notionally carries the same suffix N. However, if the suffix is not present, the application will try looking for a parameter with the same name with no suffix instead. In this way, if several layers have the same value for a given parameter (for instance input table), you can supply it using one unsuffixed parameter to save having to supply several parameters with the same value but different suffixes.

legborder = true|false ***(Boolean)***
If true, a line border is drawn around the legend.

[Default: true]

legend = true|false ***(Boolean)***
Whether to draw a legend or not. If no value is supplied, the decision is made automatically: a legend is drawn only if it would have more than one entry.

leglabelN = <text> ***(String)***
Sets the presentation label for the layer with a given suffix. This is the text which is displayed in the legend, if present. Multiple layers may use the same label, in which case they will be combined to form a single legend entry.

If no value is supplied (the default), the suffix itself is used as the label.

legopaque = true|false ***(Boolean)***
If true, the background of the legend is opaque, and the legend obscures any plot components behind it. Otherwise, it's transparent.

[Default: true]

legpos = <xfrac,yfrac> ***(double[])***
Determines the internal position of the legend on the plot. The value is a comma-separated pair of values giving the X and Y positions of the legend within the plotting bounds, so for instance "0.5,0.5" will put the legend right in the middle of the plot. If no value is supplied, the legend will appear outside the plot boundary.

legseq = <suffix>[,...] ***(String[])***
Determines which layers are represented in the legend (if present) and in which order they appear. The legend has a line for each layer label (as determined by the leglabelN parameter). If multiple layers have the same label, they will contribute to the same entry in the legend, with style icons plotted over each other. The value of this parameter is a comma-separated sequence of layer suffixes, which determines the order in which the legend entries appear. Layers with suffixes missing from this list do not show up in the legend at all.

If no value is supplied (the default), the sequence is the same as the layer plotting sequence (see seq).

minor = true|false ***(Boolean)***
If true, minor tick marks are painted along the axes as well as the major tick marks. Minor tick

marks do not have associated grid lines.

[Default: `true`]

`omode = swing|out|cgi|discard|auto` ***(PaintMode)***
Determines how the drawn plot will be output, see Section 8.5.

- `swing`: Plot will be displayed in a window on the screen. This plot is "live"; it can be resized and (except for old-style plots) navigated around with mouse actions in the same way as plots in TOPCAT.
- `out`: Plot will be written to a file given by `out` using the graphics format given by `ofmt`.
- `cgi`: Plot will be written in a way suitable for CGI use direct from a web server. The output is in the graphics format given by `ofmt`, preceded by a suitable "Content-type" declaration.
- `discard`: Plot is drawn, but discarded. There is no output.
- `auto`: Behaves as `swing` or `out` mode depending on presence of `out` parameter

[Default: `auto`]

`parallel = <int-value>` ***(Integer)***
Determines how many threads will run in parallel if animation output is being produced. Only used if the `animate` parameter is supplied. The default value is the number of processors apparently available to the JVM.

[Default: `20`]

`phi = <degrees>` ***(Double)***
First of the Euler angles, in the ZXZ sequence, defining the rotation of the plotted 3d space. Units are degrees. This is the rotation around the initial Z axis applied before the plot is viewed.

[Default: `30`]

`psi = <degrees>` ***(Double)***
Second of the Euler angles, in the ZXZ sequence, defining the rotation of the plotted 3d space. Units are degrees.

[Default: `0`]

`scale = <number>` ***(Double)***
The length of the cube sides in data coordinates. This will be determined from the data range if not supplied.

`seq = <suffix>[,...]` ***(String[])***
Contains a comma-separated list of layer suffixes to determine the order in which layers are drawn on the plot. This can affect which symbol are plotted on top of, and so potentially obscure, which other ones.

When specifying a plot, multiple layers may be specified, each introduced by a parameter `layer<N>`, where `<N>` is a different (arbitrary) suffix labelling the layer, and is appended to all the parameters specific to defining that layer.

By default the layers are drawn on the plot in the order in which the `layer*` parameters appear on the command line. However if this parameter is specified, each comma-separated element is interpreted as a layer suffix, giving the ordered list of layers to plot. Every element of the list must be a suffix with a corresponding `layer` parameter, but missing or repeated elements are allowed.

`storage = simple|memory|disk|policy|cache|basic-cache|persistent|parallel`
(DataStoreFactory)
Determines the way that data is accessed when constructing the plot. There are two main options, cached or not. If no caching is used then rows are read sequentially from the specified input table(s) every time they are required. This generally requires a small resource footprint

(though that can depend on how the table is specified) and makes sense if the data only needs to be scanned once or perhaps if the table is very large. If caching is used then the required data is read once from the specified input table(s), then prepared and cached before any plotting is performed, and plots are done using this cached data. This may use a significant amount of storage for large tables but it's usually more sensible (faster) if the data will need to be scanned multiple times. There are various options for cache storage.

The options are:

- `simple`: no caching, data read directly from input table
- `memory`: cached to memory; OutOfMemoryError possible for very large plots
- `disk`: cached to disk
- `policy`: cached using application-wide default storage policy, which is usually *adaptive* (memory/disk hybrid)
- `persistent`: cached to persistent files on disk, in the system temporary directory (defined by system property `java.io.tmpdir`). If this is used, plot data will be stored on disk in a way that means they can be re-used between STILTS invocations, so data preparation can be avoided on subsequent runs. Note however it can leave potentially large files in your temporary directory.
- `cache`: synonym for `memory` (backward compatibility)
- `basic-cache`: dumber version of `memory` (no optimisation for constant-valued columns)
- `parallel`: experimental version of memory-based cache that reads into the cache in parallel for large files. This will make the plot faster to prepare, but interaction is a bit slower and sequence-dependent attributes of the plot may not come out right. This experimental option may be withdrawn or modified in future releases.

The default value is `memory` if a live plot is being generated (`omode=swing`), since in that case the plot needs to be redrawn every time the user performs plot navigation actions or resizes the window, or if animations are being produced. Otherwise (e.g. output to a graphics file) the default is `simple`.

[Default: `simple`]

`texttype = plain|antialias|latex` ***(TextSyntax)***

Determines how to turn label text into characters on the plot. `Plain` and `Antialias` both take the text at face value, but `Antialias` smooths the characters. `LaTeX` interprets the text as LaTeX source code and typesets it accordingly.

When not using LaTeX, antialiased text usually looks nicer, but can be perceptibly slower to plot. At time of writing, on MacOS antialiased text seems to be required to stop the writing coming out upside-down for non-horizontal text (MacOS java bug).

[Default: `plain`]

`theta = <degrees>` ***(Double)***

Second of the Euler angles, in the ZXZ sequence, defining the rotation of the plotted 3d space. Units are degrees. This is the rotation towards the viewer.

[Default: `-15`]

`title = <value>` ***(String)***

Text of a title to be displayed at the top of the plot. If null, the default, no title is shown and there's more space for the graphics.

`xoff = <pixels>` ***(Double)***

Shifts the whole plot within the plotting region by the given number of pixels in the horizontal direction.

[Default: `0`]

`xpix = <int-value>` ***(Integer)***

Size of the output image in the X direction in pixels. This includes space for any axis labels,

padding and other decoration outside the plot area itself. See also `insets`.

[Default: `500`]

`yoff = <pixels>` ***(Double)***
Shifts the whole plot within the plotting region by the given number of pixels in the vertical direction.

[Default: `0`]

`ypix = <int-value>` ***(Integer)***
Size of the output image in the Y direction in pixels. This includes space for any axis labels, padding and other decoration outside the plot area itself. See also `insets`.

[Default: `400`]

`zoom = <factor>` ***(Double)***
Sets the magnification factor at which the the plotted 3D region itself is viewed, without affecting its contents. The default value is 1, which means the cube fits into the plotting space however it is rotated. Much higher zoom factors will result in parts of the plotting region and axes being drawn outside of the plotting region (so invisible).

[Default: `1`]

`zoomfactor = <number>` ***(Double)***
Sets the amount by which the plot view zooms in or out for each unit of mouse wheel movement. A value of 1 means that mouse wheel zooming has no effect. A higher value means that the mouse wheel zooms faster and a value nearer 1 means it zooms slower. Values below 1 are not permitted.

[Default: `1.2`]

B.13.2 Examples

Some examples of `plot2cube` are shown below. See Section 8.1.3 for some examples of producing **animations**, for instance of a rotating cube.

```
stilts plot2sphere in=hip_main.fits lon=radeg lat=dedeg r=plx
                   layer1=mark shading1=density densemap1=cyan-magenta
```

Plots points with RA, Dec and parallax coordinates in 3D. Density shading is used, which means you can see the lines of sight along which most objects fall, though single points are still visible. Density shading is usually a good choice if there is just one dataset, though it can get confusing with more than one.

```
stilts plot2sphere in=hip_main.fits lon=radeg lat=dedeg r=plx
                   layer1=mark shading1=density densemap1=cyan-magenta
                   cx=0 cy=0 cz=0 scale=38 texttype=antialias gridaa=true
```

The same as the previous example but with some more configuration of the axes. The data origin is placed at the centre of the visible cube (this is the position around which the cube will rotate when you drag the mouse), and the size of the cube sides in data coordinates is set explicitly.

B.14 `plot2time`: Draws a time plot

`plot2time` draws plots where the horizontal axis represents time. The time axis can be labelled in various different ways including MJD, decimal year and ISO-8601 form.

Positional coordinates are specified as `t`, `y` pairs, with an optional `ttype` specifier to indicate how the input value is to be interpreted, e.g.:

```
plot2time in1=series.fits layer1=line t1=EPOCH ttype1=MJD y1=ENERGY
```

Time values can be represented in various ways in input data, for instance as Julian Day, Modified Julian Date, decimal years since 0AD, Unix seconds, ISO-8601, or variants of some of the above with additional offsets. In some cases the input format contains enough metadata to determine how the values should be mapped to a common timescale (so for instance they can be plotted as MJD or Year/Month/Day), and in other cases they do not. For example CDF files and VOTable 1.4 files with `TIMESYS` markup contain sufficient metadata, and text inputs using the ISO-8601 format can usually be identified and understood, but there's no way to tell automatically whether a numeric column in a CSV file represents MJD, seconds since a known epoch, decimal years, or anything else. For this reason the `ttypeN` parameter is provided for all the layer types with a `tN` coordinate, as follows:

`ttypeN = DecYear|MJD|JD|Unix|Iso8601` ***(TimeMapper)***

Selects the form in which the Time value for parameter `tN` is supplied. Options are:

- `DecYear`: Years since 0 AD
- `MJD`: Modified Julian Date
- `JD`: Julian Day
- `Unix`: Seconds since midnight 1 Jan 1970
- `Iso8601`: ISO 8601 string

If left blank, a guess will be taken depending on the data type of the value supplied for the `tN` value.

This command, unlike the other `plot2*` commands at time of writing, can be used to draw *multi-zone* plots. These are plots with different panels stacked vertically so that different datasets can share the same horizontal (time) axis, but have separate vertical axes, colour maps, legends etc. The horizontal axes are always synchronized between zones. This is currently controlled with the `zoneN` parameter. For any layer with a layer suffix `N`, you can specify a zone identifier as an arbitrary string, `Z`, by supplying the parameter `zoneN=Z`. Layers with the same value of `zoneN` are plotted in the same zone, and layers with different values are plotted in different zones. If no `zoneN` is given, the layer is assigned to a single (unnamed) zone, so with no zone parameters specified all plots appear in a single zone. Parameters specific to a given zone can then be suffixed with the same `Z` zone identifier. The examples section illustrates what this looks like in practice.

Note: *The multi-zone feature is experimental. As currently implemented it lacks some features. The interface may be changed in a future version.*

Content is added to the plot by specifying one or more *plot layers* using the `layerN` parameter. The `N` part is a suffix applied to all the parameters affecting a given layer; any suffix (including the empty string) may be used. Available layers for this plot type are: `line` (Section 8.3.22), `linearfit` (Section 8.3.23), `mark` (Section 8.3.1), `fill` (Section 8.3.28), `quantile` (Section 8.3.29), `grid` (Section 8.3.27), `histogram` (Section 8.3.30), `kde` (Section 8.3.31), `knn` (Section 8.3.32), `densogram` (Section 8.3.33), `gaussian` (Section 8.3.34), `yerror` (Section 8.3.46), `spectrogram` (Section 8.3.47), `label` (Section 8.3.24), `function` (Section 8.3.35).

B.14.1 Usage

The usage of `plot2time` is

```
stilts <stilts-flags> plot2time xpix=<int-value> ypix=<int-value>
```

```
insets=<top>,<left>,<bottom>,<right>
omode=swing|out|cgi|discard|auto
storage=simple|memory|disk|policy|cache|basic-cache|persist
seq=<suffix>[,...] legend=true|false
legborder=true|false legopaque=true|false
legseq=<suffix>[,...] legpos=<xfrac,yfrac>
title=<value>
auxmap=<map-name>|<color>-<color>[-<color>...]
auxclip=<lo>,<hi> auxflip=true|false
auxquant=<number>
auxfunc=log|linear|histogram|histolog|sqrt|square|acos|cos
auxmin=<number> auxmax=<number>
auxlabel=<text> auxcrowd=<factor>
auxwidth=<pixels> auxvisible=true|false
forcebitmap=true|false compositor=0..1
animate=<table> afmt=<in-format>
astream=true|false acmd=<cmds>
parallel=<int-value> ylog=true|false
yflip=true|false tlabel=<text>
ylabel=<text> grid=true|false
tcrowd=<number> ycrowd=<number>
tformat=iso-8601|year|mjd|unix
minor=true|false
texttype=plain|antialias|latex
fontsize=<int-value>
fontstyle=standard|serif|mono
fontweight=plain|bold|italic|bold_italic
tmin=<year-or-iso8601>
tmax=<year-or-iso8601> tsub=<lo>,<hi>
ymin=<number> ymax=<number> ysub=<lo>,<hi>
navaxes=t|y|ty zoomfactor=<number>
leglabelN=<text>
layerN=<layer-type> <layerN-specific-params>
zoneN=<text>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.plot2.task.TimePlot2Task`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`acmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the animation control table as specified by parameter `animate`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`afmt = <in-format>` ***(String)***

Specifies the format of the animation control table as specified by parameter `animate`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`animate = <table>` ***(StarTable)***

If not null, this parameter causes the command to create a sequence of plots instead of just one. The parameter value is a table with one row for each frame to be produced. Columns in the table are interpreted as parameters which may take different values for each frame; the column name is the parameter name, and the value for a given frame is its value from that row. Animating like this is considerably more efficient than invoking the STILTS command in a loop.

The location of the animation control table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `afmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`astream = true|false` ***(Boolean)***

If set true, the animation control table specified by the `animate` parameter will be read as a stream. It is necessary to give the `afmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`auxclip = <lo>,<hi>` ***(Subrange)***

Defines a subrange of the colour ramp to be used for Aux shading. The value is specified as a (low,high) comma-separated pair of two numbers between 0 and 1.

If the full range `0,1` is used, the whole range of colours specified by the selected shader will be used. But if for instance a value of `0,0.5` is given, only those colours at the left hand end of the ramp will be seen.

If the null (default) value is chosen, a default clip will be used. This generally covers most or all of the range 0-1 but for colour maps which fade to white, a small proportion of the lower end may be excluded, to ensure that all the colours are visually distinguishable from a white background. This default is usually a good idea if the colour map is being used with something like a scatter plot, where markers are plotted against a white background. However, for something like a density map when the whole plotting area is tiled with colours from the map, it may be better to supply the whole range `0,1` explicitly.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxclipZ` affects only zone `Z`.

`auxcrowd = <factor>` ***(Double)***

Determines how closely the tick marks are spaced on the Aux axis, if visible. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxcrowdZ` affects only zone `Z`.

[Default: `1.0`]

`auxflip = true|false` ***(Boolean)***
If true, the colour map on the Aux axis will be reversed.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxflipZ` affects only zone `Z`.

[Default: `false`]

`auxfunc = log|linear|histogram|histolog|sqrt|square|acos|cos` ***(Scaling)***
Defines the way that values in the Aux range are mapped to the selected colour ramp.

The available options are:

- `log`: Logarithmic scaling
- `linear`: Linear scaling
- `histogram`: Scaling follows data distribution, with linear axis
- `histolog`: Scaling follows data distribution, with logarithmic axis
- `sqrt`: Square root scaling
- `square`: Square scaling
- `acos`: Arccos Scaling
- `cos`: Cos Scaling

For all these options, the full range of data values is used, and displayed on the colour bar if applicable. The `Linear`, `Log`, `Square` and `Sqrt` options just apply the named function to the full data range. The histogram options on the other hand use a scaling function that corresponds to the actual distribution of the data, so that there are about the same number of points (or pixels, or whatever is being scaled) of each colour. The histogram options are somewhat more expensive, but can be a good choice if you are exploring data whose distribution is unknown or not well-behaved over its min-max range. The `Histogram` and `HistoLog` options both assign the colours in the same way, but they display the colour ramp with linear or logarithmic annotation respectively; the `HistoLog` option also ignores non-positive values.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxfuncZ` affects only zone `Z`.

[Default: `linear`]

`auxlabel = <text>` ***(String)***
Sets the label used to annotate the aux axis, if it is visible.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxlabelZ` affects only zone `Z`.

`auxmap = <map-name>|<color>-<color>[-<color>...]` ***(Shader)***
Color map used for Aux axis shading.

A mixed bag of colour ramps are available: `inferno`, `magma`, `plasma`, `viridis`, `cividis`, `cubehelix`, `sron`, `rainbow`, `rainbow2`, `rainbow3`, `pastel`, `accent`, `gnuplot`, `gnuplot2`, `specxby`, `set1`, `paired`, `hotcold`, `rdbu`, `piyg`, `brbg`, `cyan-magenta`, `red-blue`, `brg`, `heat`, `cold`, `light`, `greyscale`, `colour`, `standard`, `bugn`, `bupu`, `orrd`, `pubu`, `purd`, `painbow`, `huecl`, `hue`, `intensity`, `rgb_red`, `rgb_green`, `rgb_blue`, `hsv_h`, `hsv_s`, `hsv_v`, `yuv_y`, `yuv_u`, `yuv_v`, `scale_hsv_s`, `scale_hsv_v`, `scale_yuv_y`, `mask`, `blacker`, `whiter`, `transparency`. *Note:* many of these, including rainbow-like ones, are frowned upon by the visualisation community.

You can also construct your own custom colour map by giving a sequence of colour names separated by minus sign ("`-`") characters. In this case the ramp is a linear interpolation between each pair of colours named, using the same syntax as when specifying a colour value. So for instance "`yellow-hotpink-#0000ff`" would shade from yellow via hot pink to blue.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxmapZ` affects only zone `Z`.

[Default: `inferno`]

`auxmax = <number>` ***(Double)***
Maximum value of the data coordinate on the Aux axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxmaxZ` affects only zone `Z`.

`auxmin = <number>` ***(Double)***
Minimum value of the data coordinate on the Aux axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxminZ` affects only zone `Z`.

`auxquant = <number>` ***(Double)***
Allows the colour map used for the Aux axis to be quantised. If an integer value N is chosen then the colour map will be viewed as N discrete evenly-spaced levels, so that only N different colours will appear in the plot. This can be used to generate a contour-like effect, and may make it easier to trace the boundaries of regions of interest by eye.

If left blank, the colour map is nominally continuous (though in practice it may be quantised to a medium-sized number like 256).

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxquantZ` affects only zone `Z`.

`auxvisible = true|false` ***(Boolean)***
Determines whether the aux axis colour ramp is displayed alongside the plot.

If not supplied (the default), the aux axis will be visible when aux shading is used in any of the plotted layers.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxvisibleZ` affects only zone `Z`.

`auxwidth = <pixels>` ***(Integer)***
Determines the lateral size of the aux colour ramp, if visible, in pixels.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `auxwidthZ` affects only zone `Z`.

[Default: `15`]

`compositor = 0..1` ***(Compositor)***
Defines how multiple overplotted partially transparent pixels are combined to form a resulting colour. The way this is used depends on the details of the specified plot.

Currently, this parameter takes a "boost" value in the range 0..1. If the value is zero, saturation semantics are used: RGB colours are added in proporition to their associated alpha value until the total alpha is saturated (reaches 1), after which additional pixels have no further effect. For larger boost values, the effect is similar, but any non-zero alpha in the output is boosted to the given minimum value. The effect of this is that even very slightly populated pixels can be visually distinguished from unpopulated ones which may not be the case for saturation composition.

[Default: `0.05`]

`fontsize = <int-value>` ***(Integer)***
Size of the text font in points.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `fontsizeZ` affects only zone `Z`.

[Default: `12`]

`fontstyle = standard|serif|mono` ***(FontType)***

Font style for text.

The available options are:

- `standard`
- `serif`
- `mono`

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `fontstyleZ` affects only zone `Z`.

[Default: `standard`]

`fontweight = plain|bold|italic|bold_italic` ***(FontWeight)***
Font weight for text.

The available options are:

- `plain`
- `bold`
- `italic`
- `bold_italic`

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `fontweightZ` affects only zone `Z`.

[Default: `plain`]

`forcebitmap = true|false` ***(Boolean)***
Affects whether rendering of the data contents of a plot (though not axis labels etc) is always done to an intermediate bitmap rather than, where possible, being painted using graphics primitives. This is a rather arcane setting that may nevertheless have noticeable effects on the appearance and size of an output graphics file, as well as plotting time. For some types of plot (e.g. `shadingN=auto` or `shadingN=density`) it will have no effect, since this kind of rendering happens in any case.

When writing to vector graphics formats (PDF and PostScript), setting it true will force the data contents to be bitmapped. This may make the output less beautiful (round markers will no longer be perfectly round), but it may result in a much smaller file if there are very many data points.

When writing to bitmapped output formats (PNG, GIF, JPEG, ...), it fixes shapes to be the same as seen on the screen rather than be rendered at the mercy of the graphics system, which sometimes introduces small distortions.

[Default: `false`]

`grid = true|false` ***(Boolean)***
If true, grid lines are drawn on the plot at positions determined by the major tick marks. If false, they are absent.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `gridZ` affects only zone `Z`.

[Default: `false`]

`insets = <top>,<left>,<bottom>,<right>` ***(Padding)***
Defines the amount of space in pixels around the actual plotting area. This space is used for axis labels, and other decorations and any left over forms an empty border.

The size and position of the actual plotting area is determined by this parameter along with `xpix` and `ypix`.

The value of this parameter is 4 comma separated integers: `<top>,<left>,<bottom>,<right>`. Any or all of these values may be left blank, in which case the corresponding margin will be

calculated automatically according to how much space is required.

`layerN = <layer-type> <layerN-specific-params>` ***(LayerType)***
Selects one of the available plot types for layerN. A plot consists of a plotting surface, set up using the various unsuffixed parameters of the plotting command, and zero or more plot layers. Each layer is introduced by a parameter with the name `layer<N>` where the suffix "`<N>`" is a label identifying the layer and is appended to all the parameter names which configure that layer. Suffixes may be any string, including the empty string.

This parameter may take one of the following values, described in more detail in Section 8.3:

- `line (Section 8.3.22)`
- `linearfit (Section 8.3.23)`
- `mark (Section 8.3.1)`
- `fill (Section 8.3.28)`
- `quantile (Section 8.3.29)`
- `grid (Section 8.3.27)`
- `histogram (Section 8.3.30)`
- `kde (Section 8.3.31)`
- `knn (Section 8.3.32)`
- `densogram (Section 8.3.33)`
- `gaussian (Section 8.3.34)`
- `yerror (Section 8.3.46)`
- `spectrogram (Section 8.3.47)`
- `label (Section 8.3.24)`
- `function (Section 8.3.35)`

Each of these layer types comes with a list of type-specific parameters to define the details of that layer, including some or all of the following groups:

- input table parameters (e.g. `inN`, `icmdN`)
- coordinate params referring to input table columns (e.g. `xN`, `yN`)
- layer style parameters (e.g. `shadingN`, `colorN`)

Every parameter notionally carries the same suffix `N`. However, if the suffix is not present, the application will try looking for a parameter with the same name with no suffix instead. In this way, if several layers have the same value for a given parameter (for instance input table), you can supply it using one unsuffixed parameter to save having to supply several parameters with the same value but different suffixes.

`legborder = true|false` ***(Boolean)***
If true, a line border is drawn around the legend.

[Default: `true`]

`legend = true|false` ***(Boolean)***
Whether to draw a legend or not. If no value is supplied, the decision is made automatically: a legend is drawn only if it would have more than one entry.

`leglabelN = <text>` ***(String)***
Sets the presentation label for the layer with a given suffix. This is the text which is displayed in the legend, if present. Multiple layers may use the same label, in which case they will be combined to form a single legend entry.

If no value is supplied (the default), the suffix itself is used as the label.

`legopaque = true|false` ***(Boolean)***
If true, the background of the legend is opaque, and the legend obscures any plot components behind it. Otherwise, it's transparent.

[Default: `true`]

`legpos = <xfrac,yfrac>` ***(double[])***
Determines the internal position of the legend on the plot. The value is a comma-separated pair of values giving the X and Y positions of the legend within the plotting bounds, so for instance "`0.5,0.5`" will put the legend right in the middle of the plot. If no value is supplied, the legend will appear outside the plot boundary.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `legposZ` affects only zone `Z`.

`legseq = <suffix>[,...]` ***(String[])***
Determines which layers are represented in the legend (if present) and in which order they appear. The legend has a line for each layer label (as determined by the `leglabelN` parameter). If multiple layers have the same label, they will contribute to the same entry in the legend, with style icons plotted over each other. The value of this parameter is a comma-separated sequence of layer suffixes, which determines the order in which the legend entries appear. Layers with suffixes missing from this list do not show up in the legend at all.

If no value is supplied (the default), the sequence is the same as the layer plotting sequence (see `seq`).

`minor = true|false` ***(Boolean)***
If true, minor tick marks are painted along the axes as well as the major tick marks. Minor tick marks do not have associated grid lines.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `minorZ` affects only zone `Z`.

[Default: `true`]

`navaxes = t|y|ty` ***(boolean[])***
Determines the axes which are affected by the interactive navigation actions (pan and zoom). The default is `t` which means that the various mouse gestures will provide panning and zooming in the Time direction only. However, if it is set to `ty` mouse actions will affect both the horizontal and vertical axes.

[Default: `t`]

`omode = swing|out|cgi|discard|auto` ***(PaintMode)***
Determines how the drawn plot will be output, see Section 8.5.

- `swing`: Plot will be displayed in a window on the screen. This plot is "live"; it can be resized and (except for old-style plots) navigated around with mouse actions in the same way as plots in TOPCAT.
- `out`: Plot will be written to a file given by `out` using the graphics format given by `ofmt`.
- `cgi`: Plot will be written in a way suitable for CGI use direct from a web server. The output is in the graphics format given by `ofmt`, preceded by a suitable "Content-type" declaration.
- `discard`: Plot is drawn, but discarded. There is no output.
- `auto`: Behaves as `swing` or `out` mode depending on presence of `out` parameter

[Default: `auto`]

`parallel = <int-value>` ***(Integer)***
Determines how many threads will run in parallel if animation output is being produced. Only used if the `animate` parameter is supplied. The default value is the number of processors apparently available to the JVM.

[Default: `20`]

`seq = <suffix>[,...]` ***(String[])***
Contains a comma-separated list of layer suffixes to determine the order in which layers are drawn on the plot. This can affect which symbol are plotted on top of, and so potentially

obscure, which other ones.

When specifying a plot, multiple layers may be specified, each introduced by a parameter `layer<N>`, where `<N>` is a different (arbitrary) suffix labelling the layer, and is appended to all the parameters specific to defining that layer.

By default the layers are drawn on the plot in the order in which the `layer*` parameters appear on the command line. However if this parameter is specified, each comma-separated element is interpreted as a layer suffix, giving the ordered list of layers to plot. Every element of the list must be a suffix with a corresponding `layer` parameter, but missing or repeated elements are allowed.

`storage = simple|memory|disk|policy|cache|basic-cache|persistent|parallel` ***(DataStoreFactory)***

Determines the way that data is accessed when constructing the plot. There are two main options, cached or not. If no caching is used then rows are read sequentially from the specified input table(s) every time they are required. This generally requires a small resource footprint (though that can depend on how the table is specified) and makes sense if the data only needs to be scanned once or perhaps if the table is very large. If caching is used then the required data is read once from the specified input table(s), then prepared and cached before any plotting is performed, and plots are done using this cached data. This may use a significant amount of storage for large tables but it's usually more sensible (faster) if the data will need to be scanned multiple times. There are various options for cache storage.

The options are:

- `simple`: no caching, data read directly from input table
- `memory`: cached to memory; OutOfMemoryError possible for very large plots
- `disk`: cached to disk
- `policy`: cached using application-wide default storage policy, which is usually *adaptive* (memory/disk hybrid)
- `persistent`: cached to persistent files on disk, in the system temporary directory (defined by system property `java.io.tmpdir`). If this is used, plot data will be stored on disk in a way that means they can be re-used between STILTS invocations, so data preparation can be avoided on subsequent runs. Note however it can leave potentially large files in your temporary directory.
- `cache`: synonym for `memory` (backward compatibility)
- `basic-cache`: dumber version of `memory` (no optimisation for constant-valued columns)
- `parallel`: experimental version of memory-based cache that reads into the cache in parallel for large files. This will make the plot faster to prepare, but interaction is a bit slower and sequence-dependent attributes of the plot may not come out right. This experimental option may be withdrawn or modified in future releases.

The default value is `memory` if a live plot is being generated (`omode=swing`), since in that case the plot needs to be redrawn every time the user performs plot navigation actions or resizes the window, or if animations are being produced. Otherwise (e.g. output to a graphics file) the default is `simple`.

[Default: `simple`]

`tcrowd = <number>` ***(Double)***

Determines how closely the tick marks are spaced on the Time axis. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `tcrowdZ` affects only zone `Z`.

[Default: `1`]

texttype = plain|antialias|latex ***(TextSyntax)***
Determines how to turn label text into characters on the plot. `Plain` and `Antialias` both take the text at face value, but `Antialias` smooths the characters. `LaTeX` interprets the text as LaTeX source code and typesets it accordingly.

When not using LaTeX, antialiased text usually looks nicer, but can be perceptibly slower to plot. At time of writing, on MacOS antialiased text seems to be required to stop the writing coming out upside-down for non-horizontal text (MacOS java bug).

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `texttypeZ` affects only zone `Z`.

[Default: `plain`]

tformat = iso-8601|year|mjd|unix ***(TimeFormat)***
Selects the way in which time values are represented when using them to label the time axis.

The available options are:

- `iso-8601`: ISO 8601 date, of the form yyyy-mm-ddThh:mm:ss.s (e.g. "2012-03-13T04")
- `year`: Decimal year (e.g. "2012.197")
- `mjd`: Modified Julian Date (e.g. "55999.2")
- `unix`: Seconds since midnight of 1 Jan 1970 (e.g. "1331613420")

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `tformatZ` affects only zone `Z`.

[Default: `iso-8601`]

title = <value> ***(String)***
Text of a title to be displayed at the top of the plot. If null, the default, no title is shown and there's more space for the graphics.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `titleZ` affects only zone `Z`.

tlabel = <text> ***(String)***
Gives a label to be used for annotating the Time axis. If not supplied no label will be drawn.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `tlabelZ` affects only zone `Z`.

tmax = <year-or-iso8601> ***(Double)***
Maximum value of the time coordinate plotted. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

The value may be set with a string that can be interpreted as a decimal year (e.g. "`2007.521`") or an ISO-8601 string (e.g. "`2007-07-10T03:57:36`", "`2007-07-10T03`" or "`2007-07-10`"). Note however that the numeric value of this configuration item if accessed programmatically is seconds since 1 Jan 1970.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `tmaxZ` affects only zone `Z`.

tmin = <year-or-iso8601> ***(Double)***
Minimum value of the time coordinate plotted. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

The value may be set with a string that can be interpreted as a decimal year (e.g. "`2007.521`") or an ISO-8601 string (e.g. "`2007-07-10T03:57:36`", "`2007-07-10T03`" or "`2007-07-10`"). Note however that the numeric value of this configuration item if accessed programmatically is seconds since 1 Jan 1970.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `tminZ`

affects only zone Z.

tsub = <lo>,<hi> ***(Subrange)***
Defines a normalised adjustment to the data range of the Time axis. The value may be specified as a comma-separated pair of two numbers, giving the lower and upper bounds of the range of of interest respectively. This sub-range is applied to the data range that would otherwise be used, either automatically calculated or explicitly supplied; zero corresponds to the lower bound and one to the upper.

The default value "0,1" therefore has no effect. The range could be restricted to its lower half with the value 0,0.5.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. tsubZ affects only zone Z.

[Default: 0,1]

xpix = <int-value> ***(Integer)***
Size of the output image in the X direction in pixels. This includes space for any axis labels, padding and other decoration outside the plot area itself. See also insets.

[Default: 500]

ycrowd = <number> ***(Double)***
Determines how closely the tick marks are spaced on the Y axis. The default value is 1, meaning normal crowding. Larger values result in more ticks, and smaller values fewer ticks. Tick marks will not however be spaced so closely that the labels overlap each other, so to get very closely spaced marks you may need to reduce the font size as well.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. ycrowdZ affects only zone Z.

[Default: 1]

yflip = true|false ***(Boolean)***
If true, the scale on the Y axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. yflipZ affects only zone Z.

[Default: false]

ylabel = <text> ***(String)***
Gives a label to be used for annotating axis Y A default value based on the plotted data will be used if no value is supplied.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. ylabelZ affects only zone Z.

[Default: Y]

ylog = true|false ***(Boolean)***
If false (the default), the scale on the Y axis is linear, if true it is logarithmic.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. ylogZ affects only zone Z.

[Default: false]

ymax = <number> ***(Double)***
Maximum value of the data coordinate on the Y axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. ymaxZ affects only zone Z.

ymin = <number> ***(Double)***

Minimum value of the data coordinate on the Y axis. This sets the value before any subranging is applied. If not supplied, the value is determined from the plotted data.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `yminZ` affects only zone `Z`.

ypix = <int-value> ***(Integer)***

Size of the output image in the Y direction in pixels. This includes space for any axis labels, padding and other decoration outside the plot area itself. See also `insets`.

[Default: `400`]

ysub = <lo>,<hi> ***(Subrange)***

Defines a normalised adjustment to the data range of the Y axis. The value may be specified as a comma-separated pair of two numbers, giving the lower and upper bounds of the range of of interest respectively. This sub-range is applied to the data range that would otherwise be used, either automatically calculated or explicitly supplied; zero corresponds to the lower bound and one to the upper.

The default value "`0,1`" therefore has no effect. The range could be restricted to its lower half with the value `0,0.5`.

If a zone suffix is appended to the parameter name, only that zone is affected, e.g. `ysubZ` affects only zone `Z`.

[Default: `0,1`]

zoneN = <text> ***(String)***

Defines which plot zone the layer with suffix N will appear in. This only makes sense for multi-zone plots. The actual value of the parameter is not significant, it just serves as a label, but different layers will end up in the same plot zone if they give the same values for this parameter.

zoomfactor = <number> ***(Double)***

Sets the amount by which the plot view zooms in or out for each unit of mouse wheel movement. A value of 1 means that mouse wheel zooming has no effect. A higher value means that the mouse wheel zooms faster and a value nearer 1 means it zooms slower. Values below 1 are not permitted.

[Default: `1.2`]

B.14.2 Examples

Here are some examples of `plot2time`:

```
stilts plot2time xpix=1000 ypix=300
                 in=ACE_data.vot t=epoch
                 layer.r=line y.r=Br color.r=grey
                 layer.t=line y.t=Bt color.t=cyan
                 layer.n=line y.n=Bn color.n=pink
```

Three time series are plotted on the same axes as lines in different colours. In this case we trust the markup in the VOTable input to describe the format of the `epoch` column.

```
stilts plot2time xpix=1000 ypix=1000
                 in=ACE_data.vot t=fp_year ttype=decyear
                 layer.r=line y.r=Br zone.r=ZR
                 layer.t=line y.t=Bt zone.t=ZT
                 layer.n=line y.n=Bn zone.n=ZN
                 titleZR="Br" titleZT="Bt" titleZN="Bn"
```

The same data is plotted as in the previous example, but in this case each line is drawn in a

different panel (zone), stacked vertically. The default colour is used for each line. Each plot is given a different title; note the `title` parameter suffixes refer to the zone identifiers not the layer identifiers. The time coordinate in this case is supplied as a decimal year column which does not have sufficient metadata to identify it as such, so we specify the time format explicitly as `decyear`.

```
stilts plot2time tmin=2007-06-07T02:40 tmax=2007-06-07T06:20 tformat=mjd
                 in=STEREO_STA_L1_SEPT_20070607_V05.cdf t=epoch_ns
                 ylabel=Channel
                 layer_3=spectrogram spectrum_3=Spec_0_NS
                 auxmap=accent auxfunc=log
```

Plots a spectrogram from a CDF file. The range along the horizontal axis is specified explicitly using ISO-8601 date strings, but it is labelled in Modified Julian Date.

```
stilts plot2time in=STEREO_STA_L1_SEPT_20070607_V05.cdf t=epoch_ns
                 layer_1=spectrogram spectrum_1=spec_0_ns zone_1=A
                 layer_2=spectrogram spectrum_2=spec_0_e  zone_2=B
                 layer_3=line y_3='mean(spec_0_ns)' color_3=plum    zone_3=C
                 layer_4=line y_4='mean(spec_0_e)'  color_4=skyblue zone_4=C
                 ylogC=true
                 auxfunc=sqrt auxmapA=viridis auxmapB=magma
```

This is a 3-zone plot; zones `A` and `B` each contains a spectrogram (layers `_1` and `_2`), and zone `C` contains two line plots (layers `_3` and `_4`). The Y axis is set to logarithmic for zone C only. The colour ramps are configured with `aux*` parameters; the stretch function is set to `sqrt` for all zones, and the colour map is set to different values for zones A and B.

B.15 `plot2d`: Old-style 2D Scatter Plot

This section describes a deprecated command. It still works, but you are advised to use the more capable `plot2plane` instead.

`plot2d` performs two-dimensional scatter plots, sending the output to a graphical display or writing it to a file in some vector or bitmapped graphics format. You need to supply it with values for one or more X and Y datasets, in terms of table columns, and it will generate a plot with a point for each row. There are many options available to configure the detailed appearance of the plot, but in its simplest form invocation is quite straightforward. See Section 9 for more discussion on use of the plotting commands.

B.15.1 Usage

The usage of `plot2d` is

```
stilts <stilts-flags> plot2d xpix=<int-value> ypix=<int-value>
                            font=dialog|serif|... fontsize=<int-value>
                            fontstyle=plain|bold|italic|bold-italic
                            legend=true|false title=<value>
                            omode=swing|out|cgi|discard|auto
                            out=<out-file>
                            ofmt=png|png-transp|gif|jpeg|pdf|svg|eps|eps-gzip
                            inN=<table> ifmtN=<in-format>
                            istreamN=true|false cmdN=<cmds> xdataN=<expr>
                            ydataN=<expr> auxdataN=<expr>
                            xlo=<float-value> ylo=<float-value>
                            auxlo=<float-value> xhi=<float-value>
                            yhi=<float-value> auxhi=<float-value>
                            xlog=true|false ylog=true|false
                            auxlog=true|false xflip=true|false
                            yflip=true|false auxflip=true|false
                            xlabel=<value> ylabel=<value> auxlabel=<value>
                            xerrorN=<expr>|[<lo-expr>],[<hi-expr>]
```

```
yerrorN=<expr>|[<lo-expr>],[<hi-expr>]
auxshader=rainbow|pastel|... txtlabelN=<value>
subsetNS=<expr> nameNS=<value>
colourNS=<rrggbb>|red|blue|...
shapeNS=filled_circle|open_circle|...
sizeNS=<int-value> transparencyNS=<int-value>
lineNS=DotToDot|LinearRegression
linewidthNS=<int-value>
dashNS=dot|dash|...|<a,b,...>
hideNS=true|false
errstyleNS=lines|capped_lines|...
grid=true|false antialias=true|false
sequence=<suffix>,<suffix>,...
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TablePlot2D`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`antialias = true|false` ***(Boolean)***
Controls whether lines are drawn using antialiasing, where applicable. If lines are drawn to a bitmapped-type graphics output format setting this parameter to true smooths the lines out by using gradations of colour for diagonal lines, and setting it false simply sets each pixel in the line to on or off. For vector-type graphics output formats, or for cases in which no diagonal lines are drawn, the setting of this parameter has no effect. Setting it true may slow the plot down slightly.

[Default: `true`]

`auxdataN = <expr>` ***(String)***
Gives a column name or expression for the aux axis data for table N. The expression is a numeric algebraic expression based on column names as described in Section 10

`auxflip = true|false` ***(Boolean)***
If set true, the scale on the aux axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

`auxhi = <float-value>` ***(Double)***
The upper limit for the plotted aux axis. If not set, a value will be chosen which is high enough to accommodate all the data.

`auxlabel = <value>` ***(String)***
Specifies a label to be used for annotating axis aux. A default values based on the plotted data will be used if no value is supplied for this parameter.

`auxlo = <float-value>` ***(Double)***
The lower limit for the plotted aux axis. If not set, a value will be chosen which is low enough to accommodate all the data.

`auxlog = true|false` ***(Boolean)***
If false (the default), the scale on the aux axis is linear; if true it is logarithmic.

[Default: `false`]

`auxshader = rainbow|pastel|...` ***(Shader)***
Determines how data from auxiliary axes will be displayed. Generally this is some kind of colour ramp. These are the available *colour fixing* options:

- `rainbow`
- `pastel`
- `standard`

- heat
- colour
- hue
- greyscale
- red-blue

and these are the available *colour modifying* options:

- hsv_h
- hsv_s
- hsv_v
- intensity
- rgb_red
- rgb_green
- rgb_blue
- yuv_y
- yuv_u
- yuv_v
- transparency

[Default: rainbow]

cmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the table. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

colourNS = <rrggbb>|red|blue|... ***(Color)***

Defines the colour of markers plotted. The value may be a 6-digit hexadecimal number giving red, green and blue intensities, e.g. "ff00ff" for magenta. Alternatively it may be the name of one of the pre-defined colours. These are currently red, blue, green, grey, magenta, cyan, orange, pink, yellow, black and white.

For most purposes, either the American or the British spelling is accepted for this parameter name.

dashNS = dot|dash|...|<a,b,...> ***(float[])***

Defines the dash style for any lines drawn in data set NS To generate a dashed line the value may be one of the named dash types:

- dot
- dash
- longdash
- dotdash

or may be a comma-separated string of on/off length values such as "4,2,8,2". A null value indicates a solid line.

Only has an effect if the lineNS parameter is set to draw lines.

errstyleNS = lines|capped_lines|... ***(ErrorRenderer)***

Defines the way in which error bars (or ellipses, or...) will be represented for data set NS if errors are being displayed. The following options are available:

- none

- `lines`
- `capped_lines`
- `caps`
- `arrows`
- `ellipse`
- `crosshair_ellipse`
- `rectangle`
- `crosshair_rectangle`
- `filled_ellipse`
- `filled_rectangle`

[Default: `lines`]

`font = dialog|serif|...` ***(String)***

Determines the font that will be used for textual annotation of the plot, including axes etc. At least the following fonts will be available:

- `serif`
- `sansserif`
- `monospaced`
- `dialog`
- `dialoginput`

as well as a range of system-dependent fonts, possibly including

- `aakar`
- `abyssinica_sil`
- `ani`
- `anjalioldlipi`
- `bitstream_charter`
- `c059`
- `cantarell`
- `cantarell_extra_bold`
- `cantarell_light`
- `cantarell_thin`
- `century_schoolbook_l`
- `chandas`
- `chilanka`
- `courier`
- `courier_10_pitch`
- `d050000l`
- `dejavu_math_tex_gyre`
- `dejavu_sans`
- `dejavu_sans_condensed`
- `dejavu_sans_light`
- `dejavu_sans_mono`
- `dejavu_serif`
- `dejavu_serif_condensed`
- `dingbats`
- `...`

[Default: `dialog`]

`fontsize = <int-value>` ***(Integer)***

Sets the font size used for plot annotations.

[Default: `12`]

`fontstyle = plain|bold|italic|bold-italic` ***(Integer)***

Gives a style in which the font is to be applied for plot annotations. Options are `plain`, `bold`, `italic` and `bold-italic`.

[Default: `plain`]

`grid = true|false` ***(Boolean)***
If true, grid lines are drawn on the plot. If false, they are absent.

[Default: `true`]

`hideNS = true|false` ***(Boolean)***
Indicates whether the actual markers plotted for each point should be hidden. Normally this is false, but you may want to set it to true if the point positions are being revealed in some other way, for instance by error markers or lines drawn between them.

[Default: `false`]

`ifmtN = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`legend = true|false` ***(Boolean)***
Determines whether a legend showing which plotting style is used for each data set. Defaults to true if there is more than one set, false otherwise.

`lineNS = DotToDot|LinearRegression` ***(Line)***
Determines what line if any will be plotted along with the data points. The options are:

- `null`: No line is plotted.
- `DotToDot`: Each point is joined to the next one in sequence by a straight line.

- `LinearRegression`: A linear regression line is plotted based on all the points which are visible in the plot. Note that the regression coefficients take no account of points out of the visible range.

`linewidthNS = <int-value>` ***(Integer)***
Sets the line width in pixels for any lines drawn in data set NS.

Only has an effect if the `lineNS` parameter is set to draw lines.

[Default: `1`]

`nameNS = <value>` ***(String)***
Provides a name to use for a subset with the symbolic label NS. This name will be used for display in the legend, if one is displayed.

`ofmt = png|png-transp|gif|jpeg|pdf|svg|eps|eps-gzip` ***(GraphicExporter)***
Graphics format in which the plot is written to the output file, see Section 8.6. One of:

- `png`: PNG
- `png-transp`: PNG with transparent background
- `gif`: GIF
- `jpeg`: JPEG
- `pdf`: Portable Document Format
- `svg`: Scalable Vector Graphics
- `eps`: Encapsulated PostScript
- `eps-gzip`: Gzipped Encapsulated PostScript

May default to a sensible value depending on the filename given by `out`.

`omode = swing|out|cgi|discard|auto` ***(PaintMode)***
Determines how the drawn plot will be output, see Section 8.5.

- `swing`: Plot will be displayed in a window on the screen. This plot is "live"; it can be resized and (except for old-style plots) navigated around with mouse actions in the same way as plots in TOPCAT.
- `out`: Plot will be written to a file given by `out` using the graphics format given by `ofmt`.
- `cgi`: Plot will be written in a way suitable for CGI use direct from a web server. The output is in the graphics format given by `ofmt`, preceded by a suitable "Content-type" declaration.
- `discard`: Plot is drawn, but discarded. There is no output.
- `auto`: Behaves as `swing` or `out` mode depending on presence of `out` parameter

[Default: `auto`]

`out = <out-file>` ***(uk.ac.starlink.util.Destination)***
The location of the output file. This is usually a filename to write to. If it is equal to the special value "-" the output will be written to standard output.

`sequence = <suffix>,<suffix>,...` ***(String[])***
Can be used to control the sequence in which different datasets and subsets are plotted. This will affect which symbols are plotted on top of, and so potentially obscure, which other ones. The value of this parameter is a comma-separated list of the "`NS`" suffixes which appear on the parameters which apply to subsets. The sets which are named will be plotted in order, so the first-named one will be at the bottom (most likely to be obscured). Note that if this parameter is supplied, then only those sets which are named will be plotted, so this parameter may also be used to restrict which plots appear (though it may not be the most efficient way of doing this). If no explicit value is supplied for this parameter, sets will be plotted in some sequence decided by STILTS (probably alphabetic by suffix).

`shapeNS = filled_circle|open_circle|...` ***(MarkShape)***
Defines the shapes for the markers that are plotted in data set NS. The following shapes are

available:

- `filled_circle`
- `open_circle`
- `cross`
- `x`
- `open_square`
- `open_diamond`
- `open_triangle_up`
- `open_triangle_down`
- `filled_square`
- `filled_diamond`
- `filled_triangle_up`
- `filled_triangle_down`

`sizeNS = <int-value>` ***(Integer)***
Defines the marker size in pixels for markers plotted in data set NS. If the value is negative, an attempt will be made to use a suitable size according to how many points there are to be plotted.

[Default: `-1`]

`subsetNS = <expr>` ***(String)***
Gives the selection criterion for the subset labelled "`NS`". This is a boolean expression which may be the name of a boolean-valued column or any other boolean-valued expression. Rows for which the expression evaluates true will be included in the subset, and those for which it evaluates false will not.

`title = <value>` ***(String)***
A one-line title to display at the top of the plot.

`transparencyNS = <int-value>` ***(Integer)***
Determines the transparency of plotted markers for data set NS. A value of `<n>` means that opacity is only achieved (the background is only blotted out) when `<n>` pixels of this colour have been plotted on top of each other.

The minimum value is 1, which means opaque markers.

`txtlabelN = <value>` ***(String)***
Gives an expression which will label each plotted point. If given, the text (or number) resulting from evaluating the expression will be written near each point which is plotted.

`xdataN = <expr>` ***(String)***
Gives a column name or expression for the x axis data for table N. The expression is a numeric algebraic expression based on column names as described in Section 10

`xerrorN = <expr>|[<lo-expr>],[<hi-expr>]` ***(String)***
Gives expressions for the errors on X coordinates for table N. The following forms are permitted:

- `<expr>`: symmetric error value
- `<lo-expr>,<hi-expr>`:distinct lower and upper error values
- `<lo-expr>,`: lower error value only
- `,<hi-expr>`: upper error value only
- `null`: no errors

The expression in each case is a numeric algebraic expression based on column names as described in Section 10.

`xflip = true|false` ***(Boolean)***
If set true, the scale on the x axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

`xhi = <float-value>` ***(Double)***
The upper limit for the plotted x axis. If not set, a value will be chosen which is high enough to accommodate all the data.

`xlabel = <value>` ***(String)***
Specifies a label to be used for annotating axis x. A default values based on the plotted data will be used if no value is supplied for this parameter.

`xlo = <float-value>` ***(Double)***
The lower limit for the plotted x axis. If not set, a value will be chosen which is low enough to accommodate all the data.

`xlog = true|false` ***(Boolean)***
If false (the default), the scale on the x axis is linear; if true it is logarithmic.

[Default: `false`]

`xpix = <int-value>` ***(Integer)***
The width of the output graphic in pixels.

[Default: `400`]

`ydataN = <expr>` ***(String)***
Gives a column name or expression for the y axis data for table N. The expression is a numeric algebraic expression based on column names as described in Section 10

`yerrorN = <expr>|[<lo-expr>],[<hi-expr>]` ***(String)***
Gives expressions for the errors on Y coordinates for table N. The following forms are permitted:

- `<expr>`: symmetric error value
- `<lo-expr>,<hi-expr>`:distinct lower and upper error values
- `<lo-expr>,`: lower error value only
- `,<hi-expr>`: upper error value only
- `null`: no errors

The expression in each case is a numeric algebraic expression based on column names as described in Section 10.

`yflip = true|false` ***(Boolean)***
If set true, the scale on the y axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

`yhi = <float-value>` ***(Double)***
The upper limit for the plotted y axis. If not set, a value will be chosen which is high enough to accommodate all the data.

`ylabel = <value>` ***(String)***
Specifies a label to be used for annotating axis y. A default values based on the plotted data will be used if no value is supplied for this parameter.

`ylo = <float-value>` ***(Double)***
The lower limit for the plotted y axis. If not set, a value will be chosen which is low enough to accommodate all the data.

`ylog = true|false` ***(Boolean)***
If false (the default), the scale on the y axis is linear; if true it is logarithmic.

[Default: `false`]

`ypix = <int-value>` ***(Integer)***
The height of the output graphic in pixels.

[Default: 300]

B.15.2 Examples

Here are some examples of plot2d in use:

```
stilts plot2d in=cat.xml xdata=RMAG-BMAG ydata=BMAG
```

Plots a colour-magnitude diagram. Since no omode or out value has been specified, the plot is posted directly to the graphics display for inspection. By adding the parameter out=xyplot.eps the plot could be written to an Encapsulated Postscript file instead.

The generated plot is here.

```
stilts plot2d in=6dfgs_mini.xml xdata=RMAG-BMAG ydata=BMAG
              subset1=SGFLAG==1 name1=galaxy colour1=blue   shape1=open_circle
              subset2=SGFLAG==2 name2=star   colour2=e010f0 shape2=x size2=3
              xlo=-1 xhi=4.5 ylo=10 yhi=20 xpix=500 ypix=250
              out=xyplot2.png
```

Plots a colour-magnitude diagram with multiple subsets. The subsets are labelled "1" and "2" with separate sets of parameters applying to each. The selections for the sets are given by the subset* parameters; set 1 is those rows with the SGFLAG column equal to 1 and set 2 is those rows with the SGFLAG column equal to 2. The boundaries of the plot in data coordinates are set explicitly rather than being determined from the data (this is faster) and the plot size in pixels is also set explicitly rather than taking the default values. Output is to a PNG file.

The generated plot is here.

```
stilts plot2d in1=iras_psc.fits cmd1='addskycoords fk5 galactic RA DEC GLON GLAT'
              xdata1=GLON ydata1=GLAT
              auxdata1=FNU_100 auxlog=true auxflip=true size1=0 transparency1=3
              in2=messier.xml   cmd2='addskycoords fk5 galactic RA DEC GLON GLAT'
              xdata2=GLON ydata2=GLAT
              txtlabel2=RADIUS>16?("M"+ID):"" cmd2='addcol SIZE sqrt(RADIUS/2)'
              xerror2=SIZE yerror2=SIZE
              subset2a=true hide2a=true colour2a=black errstyle2a=ellipse
              subset2b=true hide2b=true colour2b=black errstyle2b=filled_ellipse
                            transparency2b=6
              xlabel='Galactic Longitude' ylabel='Galactic Latitude' title='The Sky'
              legend=false grid=false fontsize=12 fontstyle=bold-italic
              xlo=0 xhi=360 ylo=-90 yhi=+90 xpix=600 ypix=300
              out=skyplot.png
```

You can do quite complicated things.

The generated plot is here.

B.16 plot3d: Old-style 3D Scatter Plot

This section describes a deprecated command. It still works, but you are advised to use the more capable plot2cube or plot2sphere instead.

plot3d performs three-dimensional scatter plots, sending the output to a graphical display or writing it to a file in some vector or bitmapped graphics format. You need to supply it with values for one or more X, Y and Z datasets, in terms of table columns, and it will generate a plot with a point for each row. There are many options available to configure the detailed appearance of the plot, but in its simplest form invocation is quite straightforward. See Section 9 for more discussion on use of the plotting commands.

B.16.1 Usage

The usage of `plot3d` is

```
   stilts <stilts-flags> plot3d xpix=<int-value> ypix=<int-value>
                               font=dialog|serif|... fontsize=<int-value>
                               fontstyle=plain|bold|italic|bold-italic
                               legend=true|false title=<value>
                               omode=swing|out|cgi|discard|auto
                               out=<out-file>
                               ofmt=png|png-transp|gif|jpeg|pdf|svg|eps|eps-gzip
                               inN=<table> ifmtN=<in-format>
                               istreamN=true|false cmdN=<cmds> xdataN=<expr>
                               ydataN=<expr> zdataN=<expr> auxdataN=<expr>
                               xlo=<float-value> ylo=<float-value>
                               zlo=<float-value> auxlo=<float-value>
                               xhi=<float-value> yhi=<float-value>
                               zhi=<float-value> auxhi=<float-value>
                               xlog=true|false ylog=true|false
                               zlog=true|false auxlog=true|false
                               xflip=true|false yflip=true|false
                               zflip=true|false auxflip=true|false
                               xlabel=<value> ylabel=<value> zlabel=<value>
                               auxlabel=<value>
                               xerrorN=<expr>|[<lo-expr>],[<hi-expr>]
                               yerrorN=<expr>|[<lo-expr>],[<hi-expr>]
                               zerrorN=<expr>|[<lo-expr>],[<hi-expr>]
                               auxshader=rainbow|pastel|... txtlabelN=<value>
                               subsetNS=<expr> nameNS=<value>
                               colourNS=<rrggbb>|red|blue|...
                               shapeNS=filled_circle|open_circle|...
                               sizeNS=<int-value> transparencyNS=<int-value>
                               lineNS=DotToDot|LinearRegression
                               linewidthNS=<int-value>
                               dashNS=dot|dash|...|<a,b,...>
                               hideNS=true|false
                               errstyleNS=lines|capped_lines|...
                               grid=true|false antialias=true|false
                               sequence=<suffix>,<suffix>,...
                               fog=<float-value> phi=<float-value>
                               theta=<float-value>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TablePlot3D`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`antialias = true|false` ***(Boolean)***
Controls whether lines are drawn using antialiasing, where applicable. If lines are drawn to a bitmapped-type graphics output format setting this parameter to true smooths the lines out by using gradations of colour for diagonal lines, and setting it false simply sets each pixel in the line to on or off. For vector-type graphics output formats, or for cases in which no diagonal lines are drawn, the setting of this parameter has no effect. Setting it true may slow the plot down slightly.

[Default: `true`]

`auxdataN = <expr>` ***(String)***
Gives a column name or expression for the aux axis data for table N. The expression is a numeric algebraic expression based on column names as described in Section 10

`auxflip = true|false` ***(Boolean)***
If set true, the scale on the aux axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

`auxhi = <float-value>` ***(Double)***
The upper limit for the plotted aux axis. If not set, a value will be chosen which is high enough to accommodate all the data.

`auxlabel = <value>` ***(String)***
Specifies a label to be used for annotating axis aux. A default values based on the plotted data will be used if no value is supplied for this parameter.

`auxlo = <float-value>` ***(Double)***
The lower limit for the plotted aux axis. If not set, a value will be chosen which is low enough to accommodate all the data.

`auxlog = true|false` ***(Boolean)***
If false (the default), the scale on the aux axis is linear; if true it is logarithmic.

[Default: `false`]

`auxshader = rainbow|pastel|...` ***(Shader)***
Determines how data from auxiliary axes will be displayed. Generally this is some kind of colour ramp. These are the available *colour fixing* options:

- `rainbow`
- `pastel`
- `standard`
- `heat`
- `colour`
- `hue`
- `greyscale`
- `red-blue`

and these are the available *colour modifying* options:

- `hsv_h`
- `hsv_s`
- `hsv_v`
- `intensity`
- `rgb_red`
- `rgb_green`
- `rgb_blue`
- `yuv_y`
- `yuv_u`
- `yuv_v`
- `transparency`

[Default: `rainbow`]

`cmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the table. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

`colourNS = <rrggbb>|red|blue|...` ***(Color)***
Defines the colour of markers plotted. The value may be a 6-digit hexadecimal number giving

red, green and blue intensities, e.g. "`ff00ff`" for magenta. Alternatively it may be the name of one of the pre-defined colours. These are currently red, blue, green, grey, magenta, cyan, orange, pink, yellow, black and white.

For most purposes, either the American or the British spelling is accepted for this parameter name.

`dashNS = dot|dash|...|<a,b,...>` ***(float[])***

Defines the dash style for any lines drawn in data set NS To generate a dashed line the value may be one of the named dash types:

- `dot`
- `dash`
- `longdash`
- `dotdash`

or may be a comma-separated string of on/off length values such as "`4,2,8,2`". A `null` value indicates a solid line.

Only has an effect if the `lineNS` parameter is set to draw lines.

`errstyleNS = lines|capped_lines|...` ***(ErrorRenderer)***

Defines the way in which error bars (or ellipses, or...) will be represented for data set NS if errors are being displayed. The following options are available:

- `none`
- `lines`
- `capped_lines`
- `caps`
- `arrows`
- `cuboid`
- `ellipse`
- `crosshair_ellipse`
- `rectangle`
- `crosshair_rectangle`
- `filled_ellipse`
- `filled_rectangle`

[Default: `lines`]

`fog = <float-value>` ***(Double)***

Sets the level of fogging used to provide a visual indication of depth. Object plotted further away from the viewer appear more washed-out by a white fog. The default value gives a bit of fogging; increase it to make the fog thicker, or set to zero if no fogging is required.

[Default: `1.0`]

`font = dialog|serif|...` ***(String)***

Determines the font that will be used for textual annotation of the plot, including axes etc. At least the following fonts will be available:

- `serif`
- `sansserif`
- `monospaced`
- `dialog`
- `dialoginput`

as well as a range of system-dependent fonts, possibly including

- `aakar`
- `abyssinica_sil`
- `ani`

- anjalioldlipi
- bitstream_charter
- c059
- cantarell
- cantarell_extra_bold
- cantarell_light
- cantarell_thin
- century_schoolbook_l
- chandas
- chilanka
- courier
- courier_10_pitch
- d050000l
- dejavu_math_tex_gyre
- dejavu_sans
- dejavu_sans_condensed
- dejavu_sans_light
- dejavu_sans_mono
- dejavu_serif
- dejavu_serif_condensed
- dingbats
- ...

[Default: dialog]

fontsize = <int-value> ***(Integer)***
Sets the font size used for plot annotations.

[Default: 12]

fontstyle = plain|bold|italic|bold-italic ***(Integer)***
Gives a style in which the font is to be applied for plot annotations. Options are plain, bold, italic and bold-italic.

[Default: plain]

grid = true|false ***(Boolean)***
If true, grid lines are drawn on the plot. If false, they are absent.

[Default: true]

hideNS = true|false ***(Boolean)***
Indicates whether the actual markers plotted for each point should be hidden. Normally this is false, but you may want to set it to true if the point positions are being revealed in some other way, for instance by error markers or lines drawn between them.

[Default: false]

ifmtN = <in-format> ***(String)***
Specifies the format of the input table as specified by parameter inN. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value (auto) (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: (auto)]

inN = <table> ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.

- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***

If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`legend = true|false` ***(Boolean)***

Determines whether a legend showing which plotting style is used for each data set. Defaults to true if there is more than one set, false otherwise.

`lineNS = DotToDot|LinearRegression` ***(Line)***

Determines what line if any will be plotted along with the data points. The options are:

- `null`: No line is plotted.
- `DotToDot`: Each point is joined to the next one in sequence by a straight line.
- `LinearRegression`: A linear regression line is plotted based on all the points which are visible in the plot. Note that the regression coefficients take no account of points out of the visible range.

`linewidthNS = <int-value>` ***(Integer)***

Sets the line width in pixels for any lines drawn in data set NS.

Only has an effect if the `lineNS` parameter is set to draw lines.

[Default: `1`]

`nameNS = <value>` ***(String)***

Provides a name to use for a subset with the symbolic label NS. This name will be used for display in the legend, if one is displayed.

`ofmt = png|png-transp|gif|jpeg|pdf|svg|eps|eps-gzip` ***(GraphicExporter)***

Graphics format in which the plot is written to the output file, see Section 8.6. One of:

- `png`: PNG
- `png-transp`: PNG with transparent background
- `gif`: GIF
- `jpeg`: JPEG
- `pdf`: Portable Document Format
- `svg`: Scalable Vector Graphics
- `eps`: Encapsulated PostScript
- `eps-gzip`: Gzipped Encapsulated PostScript

May default to a sensible value depending on the filename given by `out`.

`omode = swing|out|cgi|discard|auto` ***(PaintMode)***

Determines how the drawn plot will be output, see Section 8.5.

- `swing`: Plot will be displayed in a window on the screen. This plot is "live"; it can be resized and (except for old-style plots) navigated around with mouse actions in the same way as plots in TOPCAT.
- `out`: Plot will be written to a file given by `out` using the graphics format given by `ofmt`.
- `cgi`: Plot will be written in a way suitable for CGI use direct from a web server. The output is in the graphics format given by `ofmt`, preceded by a suitable "Content-type" declaration.
- `discard`: Plot is drawn, but discarded. There is no output.
- `auto`: Behaves as `swing` or `out` mode depending on presence of `out` parameter

[Default: `auto`]

`out = <out-file>` ***(uk.ac.starlink.util.Destination)***
The location of the output file. This is usually a filename to write to. If it is equal to the special value "-" the output will be written to standard output.

`phi = <float-value>` ***(Double)***
Angle in degrees through which the 3D plot is rotated abound the Z axis prior to drawing.

[Default: `30.0`]

`sequence = <suffix>,<suffix>,...` ***(String[])***
Can be used to control the sequence in which different datasets and subsets are plotted. This will affect which symbols are plotted on top of, and so potentially obscure, which other ones. The value of this parameter is a comma-separated list of the "`NS`" suffixes which appear on the parameters which apply to subsets. The sets which are named will be plotted in order, so the first-named one will be at the bottom (most likely to be obscured). Note that if this parameter is supplied, then only those sets which are named will be plotted, so this parameter may also be used to restrict which plots appear (though it may not be the most efficient way of doing this). If no explicit value is supplied for this parameter, sets will be plotted in some sequence decided by STILTS (probably alphabetic by suffix).

`shapeNS = filled_circle|open_circle|...` ***(MarkShape)***
Defines the shapes for the markers that are plotted in data set NS. The following shapes are available:

- `filled_circle`
- `open_circle`
- `cross`
- `x`
- `open_square`
- `open_diamond`
- `open_triangle_up`
- `open_triangle_down`
- `filled_square`
- `filled_diamond`
- `filled_triangle_up`
- `filled_triangle_down`

`sizeNS = <int-value>` ***(Integer)***
Defines the marker size in pixels for markers plotted in data set NS. If the value is negative, an attempt will be made to use a suitable size according to how many points there are to be plotted.

[Default: `-1`]

`subsetNS = <expr>` ***(String)***
Gives the selection criterion for the subset labelled "`NS`". This is a boolean expression which

may be the name of a boolean-valued column or any other boolean-valued expression. Rows for which the expression evaluates true will be included in the subset, and those for which it evaluates false will not.

`theta = <float-value>` ***(Double)***
Angle in degrees through which the 3D plot is rotated towards the viewer (i.e. about the horizontal axis of the viewing plane) prior to drawing.

[Default: `15.0`]

`title = <value>` ***(String)***
A one-line title to display at the top of the plot.

`transparencyNS = <int-value>` ***(Integer)***
Determines the transparency of plotted markers for data set NS. A value of `<n>` means that opacity is only achieved (the background is only blotted out) when `<n>` pixels of this colour have been plotted on top of each other.

The minimum value is 1, which means opaque markers.

`txtlabelN = <value>` ***(String)***
Gives an expression which will label each plotted point. If given, the text (or number) resulting from evaluating the expression will be written near each point which is plotted.

`xdataN = <expr>` ***(String)***
Gives a column name or expression for the x axis data for table N. The expression is a numeric algebraic expression based on column names as described in Section 10

`xerrorN = <expr>|[<lo-expr>],[<hi-expr>]` ***(String)***
Gives expressions for the errors on X coordinates for table N. The following forms are permitted:

- `<expr>`: symmetric error value
- `<lo-expr>,<hi-expr>`:distinct lower and upper error values
- `<lo-expr>,`: lower error value only
- `,<hi-expr>`: upper error value only
- `null`: no errors

The expression in each case is a numeric algebraic expression based on column names as described in Section 10.

`xflip = true|false` ***(Boolean)***
If set true, the scale on the x axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

`xhi = <float-value>` ***(Double)***
The upper limit for the plotted x axis. If not set, a value will be chosen which is high enough to accommodate all the data.

`xlabel = <value>` ***(String)***
Specifies a label to be used for annotating axis x. A default values based on the plotted data will be used if no value is supplied for this parameter.

`xlo = <float-value>` ***(Double)***
The lower limit for the plotted x axis. If not set, a value will be chosen which is low enough to accommodate all the data.

`xlog = true|false` ***(Boolean)***
If false (the default), the scale on the x axis is linear; if true it is logarithmic.

[Default: `false`]

`xpix = <int-value>` ***(Integer)***

The width of the output graphic in pixels.

[Default: `300`]

`ydataN = <expr>` ***(String)***
Gives a column name or expression for the y axis data for table N. The expression is a numeric algebraic expression based on column names as described in Section 10

`yerrorN = <expr>|[<lo-expr>],[<hi-expr>]` ***(String)***
Gives expressions for the errors on Y coordinates for table N. The following forms are permitted:

- `<expr>`: symmetric error value
- `<lo-expr>,<hi-expr>`:distinct lower and upper error values
- `<lo-expr>,`: lower error value only
- `,<hi-expr>`: upper error value only
- `null`: no errors

The expression in each case is a numeric algebraic expression based on column names as described in Section 10.

`yflip = true|false` ***(Boolean)***
If set true, the scale on the y axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

`yhi = <float-value>` ***(Double)***
The upper limit for the plotted y axis. If not set, a value will be chosen which is high enough to accommodate all the data.

`ylabel = <value>` ***(String)***
Specifies a label to be used for annotating axis y. A default values based on the plotted data will be used if no value is supplied for this parameter.

`ylo = <float-value>` ***(Double)***
The lower limit for the plotted y axis. If not set, a value will be chosen which is low enough to accommodate all the data.

`ylog = true|false` ***(Boolean)***
If false (the default), the scale on the y axis is linear; if true it is logarithmic.

[Default: `false`]

`ypix = <int-value>` ***(Integer)***
The height of the output graphic in pixels.

[Default: `300`]

`zdataN = <expr>` ***(String)***
Gives a column name or expression for the z axis data for table N. The expression is a numeric algebraic expression based on column names as described in Section 10

`zerrorN = <expr>|[<lo-expr>],[<hi-expr>]` ***(String)***
Gives expressions for the errors on Z coordinates for table N. The following forms are permitted:

- `<expr>`: symmetric error value
- `<lo-expr>,<hi-expr>`:distinct lower and upper error values
- `<lo-expr>,`: lower error value only
- `,<hi-expr>`: upper error value only
- `null`: no errors

The expression in each case is a numeric algebraic expression based on column names as described in Section 10.

zflip = true|false ***(Boolean)***
If set true, the scale on the z axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

zhi = <float-value> ***(Double)***
The upper limit for the plotted z axis. If not set, a value will be chosen which is high enough to accommodate all the data.

zlabel = <value> ***(String)***
Specifies a label to be used for annotating axis z. A default values based on the plotted data will be used if no value is supplied for this parameter.

zlo = <float-value> ***(Double)***
The lower limit for the plotted z axis. If not set, a value will be chosen which is low enough to accommodate all the data.

zlog = true|false ***(Boolean)***
If false (the default), the scale on the z axis is linear; if true it is logarithmic.

[Default: `false`]

B.16.2 Examples

Here are some examples of `plot3d` in use:

```
stilts plot3d in=cat.xml xdata=RMAG ydata=BMAG zdata=VEL zlog=true
```

Plots a 3-d scatter plot of red magnitude vs. blue magnitude vs. velocity; the velocity is plotted on a logarithmic scale. Since no `omode` or `out` value has been specified, the plot is posted directly to the graphics display for inspection. By adding the parameter `out=xyplot.eps` the plot could be written to an Encapsulated Postscript file instead.

The generated plot is here.

```
stilts plot3d in=sim1.fits xdata=x ydata=y zdata=z
             cmd='addcol vel "sqrt(velx*velx+vely*vely+velz*velz)"' auxdata=vel auxlog=true
             xpix=500 ypix=400 phi=50 theta=10 out=cube.jpeg
```

Plots the x, y, z positions of particles from a file containing the result of a simulation run. Here an auxiliary axis is used to colour-code the points according their velocity. This is done by introducing a new `vel` column to the table using the `addcol` filter command, so that the `vel` column can be used as the value for the `auxdata` parameter. Alternatively, the given expression for the velocity could have been used directly as the value of the `auxdata` parameter.

Additionally, the `phi` and `theta` parameters are given to adjust the orientation of the cube.

The generated plot is here.

B.17 `plothist`: Old-style Histogram

This section describes a deprecated command. It still works, but you are advised to use the more capable `plot2plane` instead.

`plothist` performs histogram plots, sending the output to a graphical display or writing it to a file in some vector or bitmapped graphics format. You need to supply it with values for one or more sets of X values, in terms of table columns, and it will bin the data and draw bars appropriately. Plot bounds, bin widths etc may be supplied expliicitly, but will be calculated from the data and set from

defaults as appropriate otherwise. There are many options available to configure the detailed appearance of the plot, but in its simplest form invocation is quite straightforward. See Section 9 for more discussion on use of the plotting commands.

B.17.1 Usage

The usage of `plothist` is

```
stilts <stilts-flags> plothist xpix=<int-value> ypix=<int-value>
                               font=dialog|serif|... fontsize=<int-value>
                               fontstyle=plain|bold|italic|bold-italic
                               legend=true|false title=<value>
                               omode=swing|out|cgi|discard|auto
                               out=<out-file>
                               ofmt=png|png-transp|gif|jpeg|pdf|svg|eps|eps-gzip
                               inN=<table> ifmtN=<in-format>
                               istreamN=true|false cmdN=<cmds>
                               xdataN=<expr> xlo=<float-value>
                               xhi=<float-value> xlog=true|false
                               xflip=true|false xlabel=<value>
                               subsetNS=<expr> nameNS=<value>
                               colourNS=<rrggbb>|red|blue|...
                               barstyleNS=fill|open|...
                               linewidthNS=<int-value>
                               dashNS=dot|dash|...|<a,b,...>
                               grid=true|false antialias=true|false
                               sequence=<suffix>,<suffix>,...
                               ylo=<float-value> yhi=<float-value>
                               ylog=true|false ylabel=<value>
                               weightN=<value> binwidth=<float-value>
                               norm=true|false cumulative=true|false
                               binbase=<float-value>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TableHistogram`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`antialias = true|false` ***(Boolean)***
Controls whether lines are drawn using antialiasing, where applicable. If lines are drawn to a bitmapped-type graphics output format setting this parameter to true smooths the lines out by using gradations of colour for diagonal lines, and setting it false simply sets each pixel in the line to on or off. For vector-type graphics output formats, or for cases in which no diagonal lines are drawn, the setting of this parameter has no effect. Setting it true may slow the plot down slightly.

[Default: `true`]

`barstyleNS = fill|open|...` ***(BarShape)***
Defines how histogram bars will be drawn for dataset NS. The options are:

- `fill`
- `open`
- `tops`
- `semi`
- `semitops`
- `spikes`
- `fillover`
- `openover`

[Default: `fill`]

`binbase = <float-value>` ***(Double)***

Adjusts the offset of the bins. By default zero (or one for logarithmic X axis) is a boundary between bins; other boundaries are defined by this and the bin width. If this value is adjusted, the lower bound of one of the bins will be set to this value, so all the bins move along by the corresponding distance.

[Default: `0.0`]

`binwidth = <float-value>` ***(Double)***

Defines the width on the X axis of histogram bins. If the X axis is logarithmic, then this is a multiplicative value.

`cmdN = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the table. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`colourNS = <rrggbb>|red|blue|...` ***(Color)***

Defines the colour of bars plotted for data set NS. The value may be a 6-digit hexadecimal number giving red, green and blue intensities, e.g. "`ff00ff`" for magenta. Alternatively it may be the name of one of the pre-defined colours. These are currently red, blue, green, grey, magenta, cyan, orange, pink, yellow, black and white.

For most purposes, either the American or the British spelling is accepted for this parameter name.

`cumulative = true|false` ***(Boolean)***

Determines whether historams are cumulative. When false (the default), the height of each bar is determined by counting the number of points which fall into the range on the X axis that it covers. When true, the height is determined by counting all the points between negative infinity and the upper bound of the range on the X axis that it covers.

[Default: `false`]

`dashNS = dot|dash|...|<a,b,...>` ***(float[])***

Defines the dashing pattern for lines drawn for dataset NS. To generate a dashed line the value may be one of the named dash types:

- `dot`
- `dash`
- `longdash`
- `dotdash`

or may be a comma-separated string of on/off length values such as "`4,2,8,2`". A `null` value indicates a solid line. Only certain bar styles are affected by the dash pattern.

`font = dialog|serif|...` ***(String)***

Determines the font that will be used for textual annotation of the plot, including axes etc. At least the following fonts will be available:

- `serif`
- `sansserif`
- `monospaced`
- `dialog`
- `dialoginput`

as well as a range of system-dependent fonts, possibly including

- `aakar`
- `abyssinica_sil`
- `ani`
- `anjalioldlipi`
- `bitstream_charter`
- `c059`
- `cantarell`
- `cantarell_extra_bold`
- `cantarell_light`
- `cantarell_thin`
- `century_schoolbook_l`
- `chandas`
- `chilanka`
- `courier`
- `courier_10_pitch`
- `d050000l`
- `dejavu_math_tex_gyre`
- `dejavu_sans`
- `dejavu_sans_condensed`
- `dejavu_sans_light`
- `dejavu_sans_mono`
- `dejavu_serif`
- `dejavu_serif_condensed`
- `dingbats`
- `...`

[Default: `dialog`]

`fontsize = <int-value>` ***(Integer)***
Sets the font size used for plot annotations.

[Default: `12`]

`fontstyle = plain|bold|italic|bold-italic` ***(Integer)***
Gives a style in which the font is to be applied for plot annotations. Options are `plain`, `bold`, `italic` and `bold-italic`.

[Default: `plain`]

`grid = true|false` ***(Boolean)***
If true, grid lines are drawn on the plot. If false, they are absent.

[Default: `true`]

`ifmtN = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <table>` ***(StarTable)***
The location of the input table. This may take one of the following forms:

- A filename.
- A URL.

- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istreamN = true|false` ***(Boolean)***
If set true, the input table specified by the `inN` parameter will be read as a stream. It is necessary to give the `ifmtN` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`legend = true|false` ***(Boolean)***
Determines whether a legend showing which plotting style is used for each data set. Defaults to true if there is more than one set, false otherwise.

`linewidthNS = <int-value>` ***(Integer)***
Defines the line width for lines drawn as part of the bars for dataset NS. Only certain bar styles are affected by the line width.

[Default: `2`]

`nameNS = <value>` ***(String)***
Provides a name to use for a subset with the symbolic label NS. This name will be used for display in the legend, if one is displayed.

`norm = true|false` ***(Boolean)***
Determines whether bin counts are normalised. If true, histogram bars are scaled such that summed height of all bars over the whole dataset is equal to one. Otherwise (the default), no scaling is done.

[Default: `false`]

`ofmt = png|png-transp|gif|jpeg|pdf|svg|eps|eps-gzip` ***(GraphicExporter)***
Graphics format in which the plot is written to the output file, see Section 8.6. One of:

- `png`: PNG
- `png-transp`: PNG with transparent background
- `gif`: GIF
- `jpeg`: JPEG
- `pdf`: Portable Document Format
- `svg`: Scalable Vector Graphics
- `eps`: Encapsulated PostScript
- `eps-gzip`: Gzipped Encapsulated PostScript

May default to a sensible value depending on the filename given by `out`.

`omode = swing|out|cgi|discard|auto` ***(PaintMode)***
Determines how the drawn plot will be output, see Section 8.5.

- `swing`: Plot will be displayed in a window on the screen. This plot is "live"; it can be resized and (except for old-style plots) navigated around with mouse actions in the same way as plots in TOPCAT.

- `out`: Plot will be written to a file given by `out` using the graphics format given by `ofmt`.
- `cgi`: Plot will be written in a way suitable for CGI use direct from a web server. The output is in the graphics format given by `ofmt`, preceded by a suitable "Content-type" declaration.
- `discard`: Plot is drawn, but discarded. There is no output.
- `auto`: Behaves as `swing` or `out` mode depending on presence of `out` parameter

[Default: `auto`]

`out = <out-file>` ***(uk.ac.starlink.util.Destination)***
The location of the output file. This is usually a filename to write to. If it is equal to the special value "-" the output will be written to standard output.

`sequence = <suffix>,<suffix>,...` ***(String[])***
Can be used to control the sequence in which different datasets and subsets are plotted. This will affect which symbols are plotted on top of, and so potentially obscure, which other ones. The value of this parameter is a comma-separated list of the "`NS`" suffixes which appear on the parameters which apply to subsets. The sets which are named will be plotted in order, so the first-named one will be at the bottom (most likely to be obscured). Note that if this parameter is supplied, then only those sets which are named will be plotted, so this parameter may also be used to restrict which plots appear (though it may not be the most efficient way of doing this). If no explicit value is supplied for this parameter, sets will be plotted in some sequence decided by STILTS (probably alphabetic by suffix).

`subsetNS = <expr>` ***(String)***
Gives the selection criterion for the subset labelled "`NS`". This is a boolean expression which may be the name of a boolean-valued column or any other boolean-valued expression. Rows for which the expression evaluates true will be included in the subset, and those for which it evaluates false will not.

`title = <value>` ***(String)***
A one-line title to display at the top of the plot.

`weightN = <value>` ***(String)***
Defines a weighting for each point accumulated to determine the height of plotted bars. If this parameter has a value other than 1 (the default) then instead of simply accumulating the number of points per bin to determine bar height, the bar height will be the sum over the weighting expression for the points in each bin. Note that with weighting, the figure drawn is no longer strictly speaking a histogram.

When weighted, bars can be of negative height. An anomaly of the plot as currently implemented is that the Y axis never descends below zero, so any such bars are currently invisible. This may be amended in a future release (contact the author to lobby for such an amendment).

[Default: `1`]

`xdataN = <expr>` ***(String)***
Gives a column name or expression for the x axis data for table N. The expression is a numeric algebraic expression based on column names as described in Section 10

`xflip = true|false` ***(Boolean)***
If set true, the scale on the x axis will increase in the opposite sense from usual (e.g. right to left rather than left to right).

[Default: `false`]

`xhi = <float-value>` ***(Double)***
The upper limit for the plotted x axis. If not set, a value will be chosen which is high enough to accommodate all the data.

`xlabel = <value>` ***(String)***

Specifies a label to be used for annotating axis x. A default values based on the plotted data will be used if no value is supplied for this parameter.

`xlo = <float-value>` ***(Double)***
The lower limit for the plotted x axis. If not set, a value will be chosen which is low enough to accommodate all the data.

`xlog = true|false` ***(Boolean)***
If false (the default), the scale on the x axis is linear; if true it is logarithmic.

[Default: `false`]

`xpix = <int-value>` ***(Integer)***
The width of the output graphic in pixels.

[Default: `400`]

`yhi = <float-value>` ***(Double)***
Upper bound for Y axis. Autogenerated from the data if not supplied.

`ylabel = <value>` ***(String)***
Specifies a label for annotating the vertical axis. A default value based on the type of histogram will be used if no value is supplied for this parameter.

[Default: `Count`]

`ylo = <float-value>` ***(Double)***
Lower bound for Y axis.

[Default: `0.0`]

`ylog = true|false` ***(Boolean)***
Whether to use a logarithmic scale for the Y axis.

[Default: `false`]

`ypix = <int-value>` ***(Integer)***
The height of the output graphic in pixels.

[Default: `300`]

B.17.2 Examples

Here are some examples of `plothist` in use:

```
stilts plothist in=cat.xml xdata=RMAG-BMAG
```

Plots a histogram of the R-B colour. The plot is displayed directly on the screen.

The generated plot is here.

```
stilts plothist in=cat.xml xdata=RMAG-BMAG ofmt=eps-gzip out=hist.eps.gz
```

Makes the same plot as the previous example, but writes it to a gzipped encapsulated postscript file instead of displaying it on the screen.

The generated plot is here.

```
stilts plothist inJ=2mass_xsc.fits xdataJ=j_m_k20fe barstyleJ=tops
               inH=2mass_xsc.fits xdataH=h_m_k20fe barstyleH=tops
               inK=2mass_xsc.fits xdataK=k_m_k20fe barstyleK=tops
               binwidth=0.1 xlo=12 xhi=16 xflip=true xlabel=Magnitude xpix=500
               out=2mass.png
```

Overplots histograms of three different columns from the same input table. These are treated as

three separate datasets which all happen to use the same input file. The different datasets are labelled "J", "H" and "K" so these suffixes appear on all the dataset-dependent parameters which are supplied. The binwidth and X range are specified explicitly rather than leaving them to be chosen automatically by examining the data.

The generated plot is here.

B.18 regquery: Queries the VO registry

regquery submits a query to the Virtual Observatory **registry** and returns the result as a table containing all the records which match the condition specified. The resulting table can be written out in any of the supported formats or otherwise processed in the usual ways. Making use of this command requires an understanding of the VOResource schema.

It is important to note that the results of this command give a very much flattened and incomplete view of the results of a full registry query. That is because the contents of an IVOA Registry (see the IVOA Resource Metadata and VOResource documents for more detail) are hierarchical and cannot be faithfully represented in a simple tabular structure. Other superior registry search clients exist; this command is just useful for viewing the results in a rather simplified way which can be represented as a table.

B.18.1 Usage

The usage of regquery is

```
stilts <stilts-flags> regquery query=<value> regurl=<url-value>
                               soapout=<out-file> ocmd=<cmds>
                               omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|
                               out=<out-table> ofmt=<out-format>
```

If you don't have the stilts script installed, write "java -jar stilts.jar" instead of "stilts" - see Section 3. The available <stilts-flags> are listed in Section 2.1. For programmatic invocation, the Task class for this command is uk.ac.starlink.ttools.task.RegQuery.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

ocmd = <cmds> ***(ProcessingStep[])***
Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ofmt = <out-format> ***(String)***
Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "(auto)" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`
- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

`out = <out-table>` ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: -]

`query = <value>` ***(String)***

Text of an ADQL WHERE clause targeted at the VOResource 1.0 schema defining which resource records you wish to retrieve from the registry. Some examples are:

- `@xsi:type like '%Organisation%'`
- `capability/@standardID = 'ivo://ivoa.net/std/ConeSearch' and title like '%SDSS%'`
- `curation/publisher like 'CDS%' and title like '%galax%'`

A full description of ADQL syntax and of the VOResource schema is well beyond the scope of this documentation, but in general you want to use `<field-name> like '<value>'` where '`%`' is a wildcard character. Logical operators `and` and `or` and parentheses can be used to group and combine expressions. To work out the various `<field-name>`s you need to look at the VOResource 1.0 schema.

`regurl = <url-value>` ***(URL)***

The URL of a SOAP endpoint which provides a VOResource1.0 IVOA registry service. Some known suitable registry endpoints at time of writing are

- `http://registry.astrogrid.org/astrogrid-registry/services/RegistryQueryv1_0`
- `http://registry.euro-vo.org/services/RegistrySearch`
- `http://vao.stsci.edu/directory/ristandardservice.asmx`

[Default:
`http://registry.astrogrid.org/astrogrid-registry/services/RegistryQueryv1_0`]

`soapout = <out-file>` ***(uk.ac.starlink.util.Destination)***
If set to a non-null value, this gives the destination for the text of the request and response SOAP messages. The special value "-" indicates standard output.

B.18.2 Examples

Here are some examples of `regquery`:

```
stilts regquery query="title like '%IRAS%'" ofmt=ascii out=iras.txt
```

Retrieves all the records in the registry whose `title` field contain the string "IRAS". The '`%`' characters function as wildcards for the ADQL `like` operator. The output is written to a local ASCII table which can be examined later.

```
stilts regquery query="capability/@standardID = 'ivo://ivoa.net/std/ConeSearch'
                       and curation/@publisher like '%astrogrid%'"
                omode=count
```

Searches for all resources which offer a cone search service and are published by AstroGrid. In this case the records are not stored, but the `omode=count` output mode counts the rows. This therefore tells you how many AstroGrid cone search services are in the registry.

```
stilts regquery query="capability/@standardID = 'ivo://ivoa.net/std/SSA'"
                ocmd="keepcols 'identifier accessUrl'"
                ofmt=ascii out=-
```

Queries the registry for all Simple Spectral Access services. The `keepcols` filter takes the result and throws away all the columns except for `identifier` and `accessUrl`, and these are written to the terminal int ASCII format.

B.19 `server`: Runs an HTTP server to perform STILTS commands

`server` runs an HTTP server which makes various elements of STILTS functionality available as HTTP services, so that they can be run by local or remote clients making HTTP requests rather than from the more usual command line interface.

When you run `server` it will start up a server which runs until it is interrupted, and write to standard output the *base URL* at which it can be accessed, for instance "`http://localhost:2112/stilts/`". If you point your browser here you will see some examples (hyperlinks to service requests) of how to use the server.

See Section 11 for more discussion of the server functionality available.

Note: The `server` command and associated servlet code are somewhat experimental. If you have requirements which are not currently provided, please contact the author for discussion.

B.19.1 Usage

The usage of `server` is

```
stilts <stilts-flags> server port=<int-value> basepath=<value>
                             tasks=<task-name> ...
```

```
tablefactory=file|dirs:...|locator:...
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.StiltsServer`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`basepath = <value>` ***(String)***
Base path on the server at which request URLs are rooted. The default is `/stilts`, which means that for instance requests to execute task `plot2d` should be directed to the URL `http://host:portnum/stilts/task/plot2d?name=value&name=value...`

[Default: `/stilts`]

`port = <int-value>` ***(Integer)***
Port number on which the server should run.

[Default: `2112`]

`tablefactory = file|dirs:...|locator:...` ***(StarTableFactory)***
This parameter determines how input table names (typically the `in` parameter of table processing commands) are used to acquire references to actual table data. The default behaviour is for input table names to be treated as filenames, in conjunction with some file type parameter. While this is usually sensible for local use, in server situations it may be inappropriate, since you don't want external users to have read access to your entire filesystem.

This parameter gives options for alternative ways of mapping table names to table data items. The currently available options are:

- `file`: default behaviour - names are treated as filenames
- `dirs:<dir>:<dir>:...`: following the "`dirs:`" prefix a list of directories is specified which will be searched for the file named. Note that the directory separator character differs between operating systems; it is a colon (":") for Unix-like OSs and a semi-colon (";") for MS Windows. If a given name is identical to the path-less filename in one of the `<dir>` directories, that file is used as the referenced table. File type information is ignored in this case, so the files must be one of the types which STILTS can autodetect, currently FITS or VOTable (FITS is more efficient). By using this option, clients can be restricted to using a fixed set of tables in a restricted part of the server's file system.
- `locator:<class-name>`: the `<class-name>` must be the name of a Java class on the classpath which implements the interface `uk.ac.starlink.ttools.task.TableLocator` and which has a no-arg constructor. An instance of this class will be used to resolve names to tables.

The usage and functionality of this parameter is experimental, and may change significantly in future releases.

[Default: `file`]

`tasks = <task-name> ...` ***(String)***
Gives a space-separated list of tasks which will be provided by the relevant endpoints of the running server. If the value is `null` then all tasks will be available. However, some tasks don't make a lot of sense to run from the server, so the default value is a somewhat restricted list. If the server is being exposed to external users, you might also want to reduce the list for security reasons. If you don't want any tasks made available, for instance if you want to run the plot service only, you can set this to the empty string.

[Default: `arrayjoin calc cdsskymatch cone coneskymatch datalinklint pixfoot pixsample plot2d plot3d plothist regquery sqlclient sqlskymatch sqlupdate`

taplint tapquery tapresume tapskymatch tcat tcatn tcopy tcube tgridmap tjoin tloop tmatch1 tmatch2 tmatchn tmulti tmultin tpipe tskymap tskymatch2 votcopy votlint xsdvalidate plot2plane plot2sky plot2cube plot2sphere plot2time]

B.19.2 Examples

Here are some examples of running the `server` command:

`stilts server`

Starts a server on the default port until it is interrupted. Most tasks are available in server mode. A message will be printed on standard output indicating the base URL at which it may be accessed, for instance "`http://localhost:2112/stilts/`".

`stilts server port=2100 basepath=tableserv`

Starts a server running on port 2100 with a given URL. The URL at which, for instance, the `plot2d` task can be executed will be "`http://` *host* `:2100/tableserv/task/plot2d`"

`stilts server tasks="plot2d plothist"`

Starts a server with a restricted list of tasks available. Only the plotting tasks `plot2d` and `plothist` will be available for execution by clients.

B.20 `sqlclient`: Executes SQL statements

`sqlclient` is a simple command-line client for use with SQL databases. One or more SQL statements can be supplied using the `sql` parameter. The result of each statement may be one or more update counts (for update-type statements) or tables (for query-type statements). Tables will be written to standard output in a format given by the `ofmt` parameter. Update results and timing information will be written to standard error.

In most cases, you will find life easier if you use either the database's own command-line or GUI client, or, if you require STILTS-type format conversion or post-processing, a `jdbc:`-format URL for the `in` parameter of the `tpipe` or `tcopy` commands (see Section 3.4 for more explanation of the latter). However, this command enables you to submit multiple queries over the same JDBC connection, including ones which do not generate a tabular result. It may be useful if a command-line client is not available to you for the database you are using.

This command can only be used if you have access to an SQL database via JDBC. The details of how to configure a JDBC connection to a database are discussed in Section 3.4 - obviously you will need a database to connect to and appropriate permissions on it as well as the relevant drivers.

This command is experimental, and it may be enhanced, renamed or withdrawn in future releases.

B.20.1 Usage

The usage of `sqlclient` is

```
stilts <stilts-flags> sqlclient db=<jdbc-url> user=<value> password=<value>
                               sql=<sql> ofmt=<out-format>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic

invocation, the Task class for this command is `uk.ac.starlink.ttools.task.SqlClient`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`db = <jdbc-url>` ***(Connection)***
URL which defines a connection to a database. This has the form `jdbc:<subprotocol>:<subname>` - the details are database- and driver-dependent. Consult Sun's JDBC documentation and that for the particular JDBC driver you are using for details. Note that the relevant driver class will need to be on your classpath and referenced in the `jdbc.drivers` system property as well for the connection to be made.

`ofmt = <out-format>` ***(String)***
Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

[Default: `text`]

`password = <value>` ***(String)***
Password for logging in to SQL database.

`sql = <sql>` ***(String)***
Text of an SQL statement for execution. This parameter may be repeated, or statements may be separated by semicolon (";") characters.

`user = <value>` ***(String)***
User name for logging in to SQL database. Defaults to the current username.

[Default: `mbt`]

B.20.2 Examples

Here are some examples of `sqlclient`:

```
stilts -classpath lib/drivers.jtds-1.1.jar \
       -Djdbc.drivers=net.sourceforge.jtds.jdbc.Driver \
       -Djava.net.preferIPv4Stack=true \
       sqlclient \
           db='jdbc:jtds:sqlserver://amenhotep:1433/twomass' \
           user='guest1' \
           ofmt=csv-nohead \
           sql='SET SHOWPLAN_TEXT ON' \
           sql='SELECT ra,dec FROM twomass_psc WHERE ra BETWEEN 21.7 AND 21.8 \
                                                AND dec BETWEEN 9.1 AND 9.12'
```

This sends two commands to a SQL Server database; the first one (SET SHOWPLAN...) sets a flag which causes the DB to return an execution plan rather than the result for subsequent queries, and the second makes the query itself. Since the password is not provided on the command line, a prompt for it will be issued before execution. The result is SQL Server's execution plan for the SELECT statement expressed as a headerless comma-separated value table sent to the terminal. CSV is chosen for the output format since it does not truncate wide columns.

B.21 `sqlskymatch`: Crossmatches table on sky position against SQL table

`sqlskymatch` resembles `coneskymatch` (Appendix B.5), but instead of sending an HTTP query to a

remote cone search service for each match (i.e. each row of the input table), it executes an SQL query directly. The query is a SELECT statement with a WHERE clause which makes restrictions on Right Ascension and Declination columns; the names of these columns must be given as parameters. The effect is that of a spatial join between a client-side table and a table stored in the database.

This command can only be used if you have access to an SQL database via JDBC. The details of how to configure a JDBC connection to a database are discussed in Section 3.4 - obviously you will need a database to connect to and appropriate read permissions on it as well as the relevant drivers.

Note: this task was known as `sqlcone` in its experimental form in STILTS v1.3.

B.21.1 Usage

The usage of `sqlskymatch` is

```
stilts <stilts-flags> sqlskymatch ifmt=<in-format> istream=true|false
                                  icmd=<cmds> ocmd=<cmds>
                                  omode=out|meta|stats|count|checksum|cgi|discard|topcat|sa
                                  out=<out-table> ofmt=<out-format>
                                  ra=<expr> dec=<expr> sr=<expr/deg>
                                  find=best|all|each usefoot=true|false
                                  footnside=<int-value>
                                  copycols=<colid-list> scorecol=<col-name>
                                  erract=abort|ignore|retry|retry<n>
                                  ostream=true|false fixcols=none|dups|all
                                  suffix0=<label> suffix1=<label>
                                  db=<jdbc-url> user=<value>
                                  password=<value> dbtable=<table-name>
                                  dbra=<sql-col> dbdec=<sql-col>
                                  dbunit=deg|rad
                                  tiling=hpx<K>|healpixnest<K>|healpixring<K>|htm<K>
                                  dbtile=<sql-col> selectcols=<sql-cols>
                                  where=<sql-condition>
                                  preparesql=true|false
                                  [in=]<table>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.SqlCone`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`copycols = <colid-list>` ***(String)***
List of columns from the input table which are to be copied to the output table. Each column identified here will be prepended to the columns of the combined output table, and its value for each row taken from the input table row which provided the parameters of the query which produced it. See Section 6.3 for list syntax. The default setting is "`*`", which means that all columns from the input table are included in the output.

[Default: `*`]

`db = <jdbc-url>` ***(Connection)***
URL which defines a connection to a database. This has the form `jdbc:<subprotocol>:<subname>` - the details are database- and driver-dependent. Consult Sun's JDBC documentation and that for the particular JDBC driver you are using for details. Note that the relevant driver class will need to be on your classpath and referenced in the `jdbc.drivers` system property as well for the connection to be made.

`dbdec = <sql-col>` ***(String)***
The name of a column in the SQL database table `dbtable` which gives the declination. Units

are given by `dbunit`.

`dbra = <sql-col>` ***(String)***
The name of a column in the SQL database table `dbtable` which gives the right ascension. Units are given by `dbunit`.

`dbtable = <table-name>` ***(String)***
The name of the table in the SQL database which provides the remote data.

`dbtile = <sql-col>` ***(String)***
The name of a column in the SQL database table `dbtable` which contains a sky tiling pixel index. The tiling scheme is given by the tiling parameter. Use of a tiling column is optional, but if present (and if the column is indexed in the database table) it may serve to speed up searches. Set to null if the database table contains no tiling column or if you do not wish to use one.

`dbunit = deg|rad` ***(AngleUnits)***
Units of the right ascension and declination columns identified in the database table. May be either deg[rees] (the default) or rad[ians].

[Default: `deg`]

`dec = <expr>` ***(String)***
Declination in degrees in the coordinate system for the position of each row of the input table. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

`erract = abort|ignore|retry|retry<n>` ***(ConeErrorPolicy)***
Determines what will happen if any of the individual cone search requests fails. By default the task aborts. That may be the best thing to do, but for unreliable or poorly implemented services you may find that some searches fail and others succeed so it can be best to continue operation in the face of a few failures. The options are:

- `abort`: Failure of any query terminates the task.
- `ignore`: Failure of a query is treated the same as a query which returns no rows.
- `retry`: Failed queries are retried until they succeed; an increasing delay is introduced for each failure. Use with care - if the failure is for some good, or at least reproducible reason this could prevent the task from ever completing.
- `retry<n>`: Failed queries are retried at most a fixed number `<n>` of times; an increasing delay is introduced for each failure. If failures persist the task terminates.

[Default: `abort`]

`find = best|all|each` ***(String)***
Determines which matches are retained.

- `best`: Only the matching query table row closest to the input table row will be output. Input table rows with no matches will be omitted. (Note this corresponds to the `best1` option in the pair matching commands, and `best1` is a permitted alias).
- `all`: All query table rows which match the input table row will be output. Input table rows with no matches will be omitted.
- `each`: There will be one output table row for each input table row. If matches are found, the closest one from the query table will be output, and in the case of no matches, the query table columns will be blank.

[Default: `all`]

`fixcols = none|dups|all` ***(Fixer)***
Determines how input columns are renamed before use in the output table. The choices are:

- `none`: columns are not renamed

- `dups`: columns which would otherwise have duplicate names in the output will be renamed to indicate which table they came from
- `all`: all columns will be renamed to indicate which table they came from

If columns are renamed, the new ones are determined by `suffix*` parameters.

[Default: `dups`]

`footnside = <int-value>` ***(Integer)***

Determines the HEALPix Nside parameter for use with the MOC footprint service. This tuning parameter determines the resolution of the footprint if available. Larger values give better resolution, hence a better chance of avoiding unnecessary queries, but processing them takes longer and retrieving and storing them is more expensive.

The value must be a power of 2, and at the time of writing, the MOC service will not supply footprints at resolutions greater than nside=512, so it should be <=512.

Only used if `usefoot=true`.

`icmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istream = true|false` ***(Boolean)***

If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed

automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`ocmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`
- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

`ostream = true|false` ***(Boolean)***

If set true, this will cause the operation to stream on output, so that the output table is built up as the results are obtained from the cone search service. The disadvantage of this is that some output modes and formats need multiple passes through the data to work, so depending on the

output destination, the operation may fail if this is set. Use with care (or be prepared for the operation to fail).

[Default: `false`]

`out = <out-table>` ***(TableConsumer)***
The location of the output table. This is usually a filename to write to. If it is equal to the special value "`-`" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `-`]

`password = <value>` ***(String)***
Password for logging in to SQL database.

`preparesql = true|false` ***(Boolean)***
If true, the JDBC connection will use `PreparedStatements` for the SQL SELECTs otherwise it will use simple `Statements`. This is a tuning parameter and affects only performance. On some database/driver combinations it's a lot faster set false (the default); on others it may be faster, who knows?

[Default: `false`]

`ra = <expr>` ***(String)***
Right ascension in degrees in the coordinate system for the position of each row of the input table. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

`scorecol = <col-name>` ***(String)***
Gives the name of a column in the output table to contain the distance between the requested central position and the actual position of the returned row. The distance returned is an angular distance in degrees. If a null value is chosen, no distance column will appear in the output table.

[Default: `Separation`]

`selectcols = <sql-cols>` ***(String)***
An SQL expression for the list of columns to be selected from the table in the database. A value of "`*`" retrieves all columns.

[Default: `*`]

`sr = <expr/deg>` ***(String)***
Expression which evaluates to the search radius in degrees for the request at each row of the input table. This will often be a constant numerical value, but may be the name or ID of a column in the input table, or a function involving one.

`suffix0 = <label>` ***(String)***
If the `fixcols` parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from the input table.

[Default: `_0`]

`suffix1 = <label>` ***(String)***
If the `fixcols` parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from the cone result table.

[Default: `_1`]

`tiling = hpx<K>|healpixnest<K>|healpixring<K>|htm<K>` ***(SkyTiling)***
Describes the sky tiling scheme that is in use. One of the following values may be used:

- `hpxK`: alias for `healpixnestK`
- `healpixnestK`: HEALPix using the Nest scheme at order `K`
- `healpixringK`: HEALPix using the Ring scheme at order `K`
- `htmK`: Hierarchical Triangular Mesh at level `K`

So for instance `hpx5` or `healpixnest5` would both indicate the HEALPix NEST tiling scheme at order 5.

At level K, there are 12*4^K HEALPix pixels, or 8*4^K HTM pixels on the sky. More information about these tiling schemes can be found at the HEALPix and HTM web sites.

`usefoot = true|false` ***(Boolean)***
Determines whether an attempt will be made to restrict searches in accordance with available footprint information. If this is set true, then before any of the per-row queries are performed, an attempt may be made to acquire footprint information about the servce. If such information can be obtained, then queries which fall outside the footprint, and hence which are known to yield no results, are skipped. This can speed up the search considerably.

Currently, the only footprints available are those provided by the CDS MOC (Multi-Order Coverage map) service, which covers VizieR and a few other cone search services.

[Default: `true`]

`user = <value>` ***(String)***
User name for logging in to SQL database. Defaults to the current username.

[Default: `mbt`]

`where = <sql-condition>` ***(String)***
An SQL expression further limiting the rows to be selected from the database. This will be combined with the constraints on position implied by the cone search centres and radii. The value of this parameter should just be a condition, it should not contain the `WHERE` keyword. A null value indicates no additional criteria.

B.21.2 Examples

Here are some examples of `sqlskymatch`:

```
stilts -classpath lib/drivers/mysql-connector-java.jar \
       -Djdbc.drivers=com.mysql.jdbc.Driver
       sqlskymatch in=messier.xml ra=RA dec=DEC sr=0.05 \
                   db='jdbc:mysql://localhost/ASTRO1' user=mbt \
                   dbtable=FIRST dbra=_RA2000 dbdec=_DE2000 \
                   out=matches.xml
```

This performs a series of SELECT statements on the table FIRST in the local MySQL database ASTRO1 to identify database objects in the region of each object represented in the VOTable `messier.xml`. The result, a join between the Messier and FIRST tables, is output as a VOTable called `matches.xml`. In this case a password has not been supplied on the command line, so if one is required it will be prompted for on the console.

B.22 `sqlupdate`: Updates values in an SQL table

`sqlupdate` updates values in an existing table in an SQL database. The rows to update are specified, as a normal SELECT statement, using the `select` parameter. Each column to update, and the value to write to it, are given using the `assign` parameter.

Why not just use the database's own UPDATE statement? In most cases, that would be a much better idea. However, using `sqlupdate` you can write values using STILTS's expression language

(Section 10), and hence take advantage of its various functions, without having to embed them into the database. SQL column names can be used as variables in these expressions, in the same way that table column names are used as variables in other commands such as `tpipe`.

This command can only be used if you have access to an SQL database via JDBC. The details of how to configure a JDBC connection to a database are discussed in Section 3.4 - obviously you will need a database to connect to and appropriate write permissions on it as well as the relevant drivers.

This is a somewhat specialised command, and several (database/driver-specific) things can go wrong with it. If you do not have a fairly good understanding of the database with which you are using it then you may run into problems (but then you'd be unlikely to have the permissions to do the updates in any case).

B.22.1 Usage

The usage of `sqlupdate` is

```
stilts <stilts-flags> sqlupdate db=<jdbc-url> user=<value> password=<value>
                                select=<select-stmt> assign=<col>=<expr>
                                progress=true|false
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.SqlUpdate`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`assign = <col>=<expr>` ***(Assignment[])***
Assigns new values for a given column. The assignment is made in the form `<colname>=<expr>` where `<colname>` is the name of a column in the SQL table and `<expr>` is the text of an expression using STILTS's expression language, as described in Section 10. SQL table column names or $ID identifiers may be used as variables in the usual way.

This parameter may be supplied more than once to effect multiple assignments, or multiple assignments may be made by separating them with semicolons in the value of this parameter.

`db = <jdbc-url>` ***(Connection)***
URL which defines a connection to a database. This has the form `jdbc:<subprotocol>:<subname>` - the details are database- and driver-dependent. Consult Sun's JDBC documentation and that for the particular JDBC driver you are using for details. Note that the relevant driver class will need to be on your classpath and referenced in the `jdbc.drivers` system property as well for the connection to be made.

`password = <value>` ***(String)***
Password for logging in to SQL database.

`progress = true|false` ***(Boolean)***
If true, a spinner will be drawn on standard error which shows how many rows have been updated so far.

[Default: `true`]

`select = <select-stmt>` ***(String)***
Gives the full text (including "`SELECT`") of the SELECT statement to identify which rows undergo updates.

`user = <value>` ***(String)***
User name for logging in to SQL database. Defaults to the current username.

[Default: mbt]

B.22.2 Examples

Here are some examples of sqlupdate:

```
stilts -classpath lib/drivers/mysql-connector-java.jar \
       -Djdbc.drivers=com.mysql.jdbc.Driver \
       sqlupdate db='jdbc:mysql://localhost/RADIO' user=root
       select='SELECT * from FIRST" \
       assign='HTMID=htmIndex(20,POS_EQ_RA,POS_EQ_DEC)'
```

Fills in the HTMID column of a table called FIRST in the local MySQL database RADIO, using HTM pixel indices based on the existing right ascension and declination columns in that table. The HTMID column must exist prior to executing this command.

B.23 taplint: Tests TAP services

taplint runs a series of tests on a Table Access Protocol (TAP) service and reports the results. Unlike most of the other tools in this package it is not likely to be of use to normal users; its intended use is for people developing or operating TAP services to assess their services, perhaps with a view to improving compliance.

Testing takes place in a number of stages; it is possible to choose which stages are run in by using the stages parameter. The default output (format=text) is line-based text to standard output, and each report line is of the (fairly greppable) form:

```
T-SSS-MMMMxN aaaaa...
```

where the parts have the following meanings:

- T: Report type, one of E(rror), W(arning), I(nfo), S(ummary), F(ailure). See the documentation of the report parameter for further description of what these mean. The report parameter can be used to suppress some of these; only E indicates actual service compliance errors, but including the others may make it easier to see what's going on.
- SSS: Stage abbreviation, as used in the stages parameter. The stages parameter can be used to select which stages are run.
- MMMM: Message label, which is always the same for messages generated by the same test, is usually different for messages generated by different tests, and may be somewhat mnemonic.
- x: Continuation indicator, either "-" or "+". In most cases it is "-", indicating the first line of a message, but multi-line messages (rare) use "-" for the first line and "+" for any continuation lines.
- N: Sequence number, which is 1 for the first time message T-SSS-MMMM is reported, and increases by one for each subsequent appearance. After a certain maximum (determined by the maxrepeat parameter) additional reports with the same code are no longer output individually, but a summary of the number of reports so discarded is written at the end of the section with the character "x" instead of the sequence number. This behaviour prevents the output being swamped by multiple reports of the same issue. If the maxrepeat parameter is increased above 9, more than one digit will be used here (so e.g. for maxrepeat=999, the format would be NNN not N).
- aaaaa...: Message text, a free text description of what is being reported.

If you don't like that format, others may be selected using the format parameter, which currently also supports JSON. For more flexible interaction with the output you can invoke taplint programmatically and supply your own OutputReporter instance.

TAP is a complicated beast, referencing many standards (including TAP, UWS, VODataService, ADQL, VOResource, VOSI, TAPRegExt, DALI, ObsCore, ObsLocTAP, EPN-TAP, VOTable, UCD, VOUnits, SSO, SoftID, HTTP, RDFa Lite), and it is hard to write a validator which is comprehensive, especially one which can provide useful output for services with a range of compliance levels. This tool tries to make a wide range of tests, but does not claim to be comprehensive. An idea of what tests it does perform can be gained from the stages listed in the description of the `stages` parameter. It does make a fairly good job of checking that declared metadata is consistent and matches the data actually returned from queries, it tests job submission in most of the various ways permitted by the TAP standard, and it checks all returned VOTables by effectively running them through `votlint`. Things it does not test much include complex ADQL queries, coordinate/STC-related data types, queries in non-ADQL languages, and service registration.

HTTP connections made by this validator are flagged in the `User-Agent` field with the token "`(IVOA-test)`".

B.23.1 Usage

The usage of `taplint` is

```
stilts <stilts-flags> taplint stages=[+/-]XXX ... maxtable=<int-value>
                              tables=<value> format=text|json
                              report=[EWISF]+ maxrepeat=<int-value>
                              truncate=<int-value> debug=true|false
                              interface=tap1.0|tap1.1|cap
                              syncurl=<url-value> asyncurl=<url-value>
                              tablesurl=<url-value>
                              capabilitiesurl=<url-value>
                              availabilityurl=<url-value>
                              examplesurl=<url-value>
                              [tapurl=]<url-value>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TapLint`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`asyncurl = <url-value>` ***(URL)***
Sets the URL to use for the async endpoint of the TAP service. The default value would be `<tapurl>/async`, but it may be influenced by the chosen `interface` value, and it can be further overridden by setting this value.

`availabilityurl = <url-value>` ***(URL)***
Sets the URL to use for the availability endpoint of the TAP service. The default value would be `<tapurl>/availability`, but it may be influenced by the chosen `interface` value, and it can be further overridden by setting this value.

`capabilitiesurl = <url-value>` ***(URL)***
Sets the URL to use for the capabilities endpoint of the TAP service. The default value would be `<tapurl>/capabilities`, but it may be influenced by the chosen `interface` value, and it can be further overridden by setting this value.

`debug = true|false` ***(Boolean)***
If true, debugging output including stack traces will be output along with the normal validation messages.

[Default: `false`]

`examplesurl = <url-value>` ***(URL)***
Sets the URL to use for the examples endpoint of the TAP service. The default value would be `<tapurl>/examples`, but it may be influenced by the chosen `interface` value, and it can be further overridden by setting this value.

`format = text|json` ***(OutputReporter)***
Determines the format of the output. Possible values are `text`, `json`.

Note not all of the other parameters may be applicable to all output formats.

[Default: `text`]

`interface = tap1.0|tap1.1|cap` ***(String)***
Defines how the service endpoints and the version of the TAP protocol to use for queries is determined. This may take one of the following (case-insensitive) values:

- `TAP1.0`: The standard TAP endpoints are used, without examining the service's capabilities document. The service is queried using version 1.0 of the TAP protocol.
- `TAP1.1`: The standard TAP endpoints are used, without examining the service's capabilities document. The service is queried using version 1.1 of the TAP protocol.
- `cap`: The service's capabilities document is examined, and the endpoints listed there are used.

The capabilities document, if used, is read from `tapurl`/capabilities unless the `capabilitiesurl` parameter is defined, in which case that is used.

The baseline value of all the TAP service endpoints (`sync`, `async`, `tables`, `capabilities`, `examples`) are determined by this parameter, but each of those endpoint values may be overridden individually by other endpoint-specific parameters (`syncurl`, `asyncurl`, `tablesurl`, `capabilitiesurl`, `availabilityurl`, `examplesurl`)

For default (unauthenticated) access, the default value is usually suitable.

[Default: `cap`]

`maxrepeat = <int-value>` ***(Integer)***
Puts a limit on the number of times that a single message will be repeated. By setting this to some reasonably small number, you can ensure that the output does not get cluttered up by millions of repetitions of essentially the same error.

[Default: `9`]

`maxtable = <int-value>` ***(Integer)***
Limits the number of tables from the service that will be tested. Currently, this only affects stage `MDQ`. If the value is left blank (the default), or if it is larger than the number of tables actually present in the service, it will have no effect.

`report = [EWISF]+` ***(String)***
Letters indicating which message types should be listed. Each character of the string is one of the letters `E`, `W`, `I`, `S`, `F` with the following meanings:

- `E`: Error in operation or standard compliance of the service.
- `W`: Warning that service behaviour is questionable, or contravenes a standard recommendation, but is not in actual violation of the standard.
- `I`: Information about progress, for instance details of queries made.
- `S`: Summary of previous successful/unsuccessful reports.
- `F`: Failure of the validator to perform some testing. The cause is either some error internal to the validator, or some error or missing functionality in the service which has already been reported.

[Default: `EWISF`]

`stages = [+/-]XXX ...` ***(String[])***

Determines the validation stages which the validator will peform. Each stage is represented by a short code, as follows:

- `TMV`: Validate table metadata against XML schema
- `TME`: Check content of tables metadata from /tables
- `TMS`: Check content of tables metadata from TAP_SCHEMA
- `TMC`: Compare table metadata from /tables and TAP_SCHEMA
- `UUC`: Check column units and UCDs are legal
- `CPV`: Validate capabilities against XML schema
- `CAP`: Check TAP and TAPRegExt content of capabilities document
- `AVV`: Validate availability against XML schema
- `QGE`: Make ADQL queries in sync GET mode
- `QPO`: Make ADQL queries in sync POST mode
- `QAS`: Make ADQL queries in async mode
- `UWS`: Test asynchronous UWS/TAP behaviour
- `MDQ`: Check table query result columns against declared metadata
- `OBS`: Test implementation of ObsCore Data Model
- `LOC`: Test implementation of ObsLocTAP Data Model
- `EPN`: Test implementation of EPN-TAP tables (off)
- `UPL`: Make queries with table uploads
- `EXA`: Check content of examples document

This parameter can specify what stages to run in the following ways:

- if left blank, the default list of stages (which is most or all of them) will be run
- if the value is a space-separated list of three-letter codes, it lists the stages that will be run
- if the value is a space separated list of three-letter codes preceded by a "+" or "-" character, the named stages will be removed or added to the default list

So "`TME CAP`" will run only Table Metadata and Capability stages, while "`-EXA -UPL`" will run all the default stages apart from Examples and Upload. The order in which stages are listed is not significant.

Note that removing some stages may affect the operation of others; for instance table metadata is acquired from the metadata stages, and avoiding those will mean that later stages which use the table metadata to pose queries will not be able to do so with knowledge of the database schema.

`syncurl = <url-value>` ***(URL)***
Sets the URL to use for the sync endpoint of the TAP service. The default value would be `<tapurl>/sync`, but it may be influenced by the chosen `interface` value, and it can be further overridden by setting this value.

`tables = <value>` ***(String)***
If supplied, this specifies a list of tables to test. It may be set to a space- or comma-separated list of table names for consideration; any tables not covered by this list are mostly ignored for the purposes of reporting. Matching against table names is case-insensitive, and the asterisk character "`*`" may be used as a wildcard to match any sequence of characters. Note that matching is against the declared table name which may or may not include a schema name prefix depending on service behaviour.

By default this parameter is not set, which means that all tables are considered.

`tablesurl = <url-value>` ***(URL)***
Sets the URL to use for the tables endpoint of the TAP service. The default value would be `<tapurl>/tables`, but it may be influenced by the chosen `interface` value, and it can be further overridden by setting this value.

`tapurl = <url-value>` ***(URL)***
The base URL of a Table Access Protocol service. This is the bare URL without a trailing

"/[a]sync".

In the usual case, the default values of the various endpoints (sync and async query submission, tables metadata, service-provided examples etc) use this URL as a parent and append standard sub-paths.

In some cases however, determination of the endpoints is more complicated, as determined by the `interface` parameter which may cause endpoints to be read from the capabilities document at `tapurl/capabilities`, and by other endpoint-specific parameters (`syncurl`, `asyncurl`, `tablesurl`, `capabilitiesurl`, `availabilityurl`, `examplesurl`) for fine tuning.

`truncate = <int-value>` ***(Integer)***
Limits the line length written to the output.

[Default: `640`]

B.23.2 Examples

Here are some examples of `taplint`:

`stilts taplint http://dc.g-vo.org/tap`

Performs a default validation run against the TAP service based at the given URL.

```
stilts taplint tapurl=http://gaia.esac.esa.int/tap-server/tap
               examplesurl=file://localhost/tmp/examples.xml
               stages='TME CAP EXA'
```

Executes the `EXA`mples stage against the GACS TAP service at ESAC. Most of the service endpoints (tables, capabilities, availability etc) are found at their default locations relative to the given `tapurl`. However, the Examples document is loaded instead from the local file at the URL that has been specified by the `examplesurl` parameter. This makes is possible to test a non-deployed examples document against a deployed TAP service. It may also be used if certain capabilities have been deployed at non-default locations to satisfy multiple security models or for other reasons. You can play similar tricks with the other `*url` parameters like `capabilitiesurl` and `tablesurl`, as listed in the documentation.

The `TME` (table metadata) and `CAP` (service capabilities) stages have been executed along with `EXA`, since taplint needs to pick up the metadata and capabilities in order to be able to do some of the checks on the examples it finds. If those stage names are not included in the `stages` parameter, the output will include some messages noting that fact, and the tests will be less rigorous.

```
stilts taplint tapurl=http://example.com/tap
               report=EW stages='TMS UWS' truncate=80 maxrepeat=4
```

A validation run is done against the named TAP service. Only Error and Warning type messages are output, only two validation stages are performed, lines are truncated to a maximum of 80 characters, and each message is repeated a maximum of 4 times. An invocation like this may be suitable if you find the default operation too verbose.

The output of this invocation might look like this:

```
    Section TMS: Check content of tables metadata from TAP_SCHEMA
E-TMS-CINT-1 Column principal in TAP_SCHEMA.columns has wrong type char not int
E-TMS-CINT-2 Column std in TAP_SCHEMA.columns has wrong type char not int
W-TMS-CLUN-1 Unused entry in TAP_SCHEMA.columns table: ivoa.obscore

Section UWS: Test asynchronous UWS/TAP behaviour
E-UWS-GMIM-1 Incorrect Content-Type text/xml != text/plain for http://exampl....
E-UWS-GMIM-2 Incorrect Content-Type text/xml != text/plain for http://exampl....
```

```
E-UWS-GMIM-3 Incorrect Content-Type text/xml != text/plain for http://exampl....
E-UWS-GMIM-4 Incorrect Content-Type text/xml != text/plain for http://exampl....
E-UWS-GMIM-x (3 more)

Totals: Errors: 9; Warnings: 1
```

B.24 `tapquery`: Queries a Table Access Protocol server

`tapquery` can query remote databases using the Table Access Protocol (TAP) services by submitting Astronomical Data Query Language queries to them and retrieving the results. TAP and ADQL are Virtual Observatory protocols.

Queries can be submitted in either synchronous or asynchronous mode, as determined by the `sync` parameter. In asynchronous mode, if the query has not been deleted by the time the command exits (see the `delete` parameter), the result can be picked up at a later stage using the `tapresume` command. Table uploads are supported, so it is possible (if the service supports this functionality), to upload a local table to the remote database, perform a query involving it, such as a join with a remote table of some sort, and receive the result. This powerful facility gives you crossmatches between local and remote tables.

This command does not provide any facility for querying the service for either table or capability metadata, so you will need to know about the service capabilities and database structure from some other source (possibly TOPCAT).

Note: this command has been introduced at STILTS version 2.3, at which time most available TAP services are quite new and may not fully conform to the standards, and usage patterns are still settling down. For this reason you may find that some TAP services do not behave quite as expected; it is also possible that in future versions the command behaviour or parameters will change in line with changing service profiles or in the light of user experience.

B.24.1 Usage

The usage of `tapquery` is

```
stilts <stilts-flags> tapquery nupload=<count> ufmtN=<in-format>
                               uploadN=<tableN> ucmdN=<cmds> ocmd=<cmds>
                               omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|
                               out=<out-table> ofmt=<out-format>
                               upnameN=<adql-identifier> tapurl=<url-value>
                               interface=tap1.0|tap1.1|cap
                               adql=<query-text> parse=true|false
                               sync=true|false maxrec=<nrow>
                               destruction=<iso8601>
                               executionduration=<seconds>
                               compress=true|false
                               upvotformat=TABLEDATA|BINARY|BINARY2
                               language=<lang-name> poll=<millisec>
                               progress=true|false
                               delete=finished|never|always|now
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TapQuerier`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`adql = <query-text>` ***(String)***

Astronomical Data Query Language string specifying the TAP query to execute. ADQL/S resembles SQL, so this string will likely start with "SELECT".

`compress = true|false` ***(Boolean)***

If true, the service is requested to provide HTTP-level compression for the response stream (Accept-Encoding header is set to "`gzip`", see RFC 2616). This does not guarantee that compression will happen but if the service honours this request it may result in a smaller amount of network traffic at the expense of more processing on the server and client.

[Default: `true`]

`delete = finished|never|always|now` ***(DeleteMode)***

Determines under what circumstances the UWS job is to be deleted from the server when its data is no longer required. If it is not deleted, then the job is left on the TAP server and it can be accessed via the normal UWS REST endpoints or using `tapresume` until it is destroyed by the server.

Possible values:

- `finished`: delete only if the job finished, successfully or not
- `never`: do not delete
- `always`: delete on command exit
- `now`: delete and return immediately

[Default: `finished`]

`destruction = <iso8601>` ***(String)***

Posts an updated value of the UWS DESTRUCTION parameter to the query job before it starts. This only makes sense for asynchronous jobs (`sync`=false).

The supplied value should be an ISO-8601-like string, giving the new requested job destruction time. The service is not obliged to honour this request. See UWS v1.0, sec 2.2.3.3.

`executionduration = <seconds>` ***(Long)***

Posts an updated value of the UWS EXECUTIONDURATION parameter to the query job before it starts. This only makes sense for asynchronous jobs (`sync`=false).

The supplied value is an integer giving the maximum number of wall-clock seconds for which the job is permitted to execute before being forcibly terminated. A value of zero indicates unlimited duration. The service is not obliged to honour this request. See UWS v1.0, sec 2.2.3.4.

`interface = tap1.0|tap1.1|cap` ***(String)***

Defines how the service endpoints and the version of the TAP protocol to use for queries is determined. This may take one of the following (case-insensitive) values:

- `TAP1.0`: The standard TAP endpoints are used, without examining the service's capabilities document. The service is queried using version 1.0 of the TAP protocol.
- `TAP1.1`: The standard TAP endpoints are used, without examining the service's capabilities document. The service is queried using version 1.1 of the TAP protocol.
- `cap`: The service's capabilities document is examined, and the endpoints listed there are used.

The capabilities document, if used, is read from `tapurl`/capabilities unless the `capabilitiesurl` parameter is defined, in which case that is used.

The baseline value of all the TAP service endpoints (`sync`, `async`, `tables`, `capabilities`, `examples`) are determined by this parameter, but each of those endpoint values may be overridden individually by other endpoint-specific parameters (`syncurl`, `asyncurl`, `tablesurl`, `capabilitiesurl`, `availabilityurl`, `examplesurl`)

For default (unauthenticated) access, the default value is usually suitable.

[Default: `tap1.0`]

`language = <lang-name>` ***(String)***
Language to use for the ADQL-like query. This will usually be "ADQL" (the default), but may be set to some other value supported by the service, for instance a variant indicating a different ADQL version. Note that at present, setting it to "PQL" is not sufficient to submit a PQL query.

[Default: `ADQL`]

`maxrec = <nrow>` ***(Long)***
Sets the requested maximum row count for the result of the query. The service is not obliged to respect this, but in the case that it has a default maximum record count, setting this value may raise the limit. If no value is set, the service's default policy will be used.

`nuploa d = <count>` ***(Integer)***
The number of upload tables for this task. For each of the upload tables N there will be associated parameters `ufmtN`, `uploadN` and `ucmdN`.

[Default: `0`]

`ocmd = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***
Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui`
(ProcessingMode)
The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`

- discard
- topcat
- samp
- plastic
- tosql
- gui

Use the help=omode flag or see Section 6.4 for more information.

[Default: out]

out = <out-table> ***(TableConsumer)***
The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if omode has its default value of "out".

[Default: -]

parse = true|false ***(Boolean)***
Determines whether an attempt will be made to check the syntax of the ADQL prior to submitting the query. If this is set true, and if a syntax error is found, the task will fail with an error before any attempt is made to submit the query.

[Default: false]

poll = <millisec> ***(Integer)***
Interval to wait between polling attempts, in milliseconds. Asynchronous TAP queries can only find out when they are complete by repeatedly polling the server to find out the job's status. This parameter allows you to set how often that happens. Attempts to set it too low (<50) will be rejected on the assumption that you're thinking in seconds.

[Default: 5000]

progress = true|false ***(Boolean)***
If this parameter is set true, progress of the job is reported to standard output as it happens.

[Default: true]

sync = true|false ***(Boolean)***
Determines whether the TAP query is submitted in synchronous or asynchronous mode. Synchronous (true) means that the result is retrieved over the same HTTP connection that the query is submitted from. This is uncomplicated, but means if the query takes a long time it may time out and the results will be lost. Asynchronous (false) means that the job is queued and results may be retrieved later. Normally this command does the necessary waiting around and recovery of the result, though with appropriate settings you can get tapresume to pick it up for you later instead. In most cases false (the default) is preferred.

[Default: false]

tapurl = <url-value> ***(URL)***
The base URL of a Table Access Protocol service. This is the bare URL without a trailing "/[a]sync".

In the usual case, the default values of the various endpoints (sync and async query submission, tables metadata, service-provided examples etc) use this URL as a parent and append standard sub-paths.

In some cases however, determination of the endpoints is more complicated, as determined by the interface parameter which may cause endpoints to be read from the capabilities document at tapurl/capabilities, and by other endpoint-specific parameters (syncurl, asyncurl, tablesurl, capabilitiesurl, availabilityurl, examplesurl) for fine tuning.

ucmdN = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on upload table #N as specified by parameter `uploadN`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ufmtN = <in-format>` ***(String)***

Specifies the format of upload table #N as specified by parameter `uploadN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`uploadN = <tableN>` ***(StarTable)***

The location of upload table #N. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ufmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`upnameN = <adql-identifier>` ***(String)***

Identifier to use in server-side expressions for uploaded table #N. In ADQL expressions, the table should be referred to as "`TAP_UPLOAD.<label>`".

The value must syntactically be an ADQL identifier (`[A-Za-z][A-Za-z0-9_]*`).

[Default: `upN`]

`upvotformat = TABLEDATA|BINARY|BINARY2` ***(uk.ac.starlink.votable.VOTableWriter)***

Determines how any uploaded tables will be serialized for transmission to the TAP server. The supplied string is the name of one of the defined VOTable serialization formats. The choice shouldn't affect any results, though it may affect required bandwidth, and some services may (though should not) have non-standard requirements for serialization format.

[Default: `TABLEDATA`]

B.24.2 Examples

Here are some examples of `tapquery`:

```
stilts tapquery tapurl='http://dc.zah.uni-heidelberg.de/__system__/tap/run/tap'
               adql='SELECT TOP 1000 * FROM ppmxl.main'
```

```
                out=ppmxl.fits
```

Executes the given ADQL query on the service referenced by the URL and writes the result to a FITS file.

```
stilts tapquery
       tapurl='http://dc.zah.uni-heidelberg.de/__system__/tap/run/tap'
       adql="SELECT *
                FROM twomass.data AS t
                JOIN TAP_UPLOAD.up1 AS s
                ON 1=CONTAINS(POINT('ICRS', t.RAJ2000, t.DEJ2000),
                              CIRCLE('ICRS', s.ra2000, s.dec2000, 5./3600.))"
       nupload=1 upload1=6dfgs_E7.fits ucmd1='select BMAG-RMAG<0'
       maxrec=20000
       ocmd='tablename 2mass_x_6df' omode=topcat
```

The local table `6dfgs_E7` is filtered to contain only rather blue objects, and the resulting selection is uploaded to the TAP server. A positional crossmatch with 5 arcsec tolerance is then performed on the server between this uploaded table and the `twomass.data` table held by the service. The adjusted `maxrec` parameter ensures that the result will not be artificially truncated to shorter than 20000 rows (assuming the service limits permit this). When the result is received, it is loaded directly into TOPCAT with the name "2mass_x_6df".

B.25 `tapresume`: Resumes a previous query to a Table Access Protocol server

`tapresume` can resume monitoring and data retrieval from an asynchronous Table Access Protocol query which has already been submitted. TAP is a Virtual Observatory protocol. Such a pre-existing query may have been submitted by the `tapquery` command or by some completely different mechanism. It essentially does the same job as `tapquery` but without the job submission stage. It waits until the query has completed, and then retrieves the table result and processes it in accordance with the supplied parameters. The query may or may not be deleted from the server as part of the operation.

B.25.1 Usage

The usage of `tapresume` is

```
stilts <stilts-flags> tapresume joburl=<url-value> compress=true|false
                                poll=<millisec> progress=true|false
                                delete=finished|never|always|now
                                ocmd=<cmds>
                                omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp
                                out=<out-table> ofmt=<out-format>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TapResume`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`compress = true|false` ***(Boolean)***

If true, the service is requested to provide HTTP-level compression for the response stream (Accept-Encoding header is set to "`gzip`", see RFC 2616). This does not guarantee that compression will happen but if the service honours this request it may result in a smaller amount of network traffic at the expense of more processing on the server and client.

[Default: `true`]

`delete = finished|never|always|now` ***(DeleteMode)***

Determines under what circumstances the UWS job is to be deleted from the server when its data is no longer required. If it is not deleted, then the job is left on the TAP server and it can be accessed via the normal UWS REST endpoints or using `tapresume` until it is destroyed by the server.

Possible values:

- `finished`: delete only if the job finished, successfully or not
- `never`: do not delete
- `always`: delete on command exit
- `now`: delete and return immediately

[Default: `finished`]

`joburl = <url-value>` ***(URL)***

The URL of a job created by submission of a TAP query which was created earlier and has not yet been deleted (by the client) or destroyed (by the server). This will usually be of the form `<tap-url>/async/<job-id>`. You can also find out, and possibly retrieve results from the job by pointing a web browser at this URL.

`ocmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`

- discard
- topcat
- samp
- plastic
- tosql
- gui

Use the help=omode flag or see Section 6.4 for more information.

[Default: out]

out = <out-table> ***(TableConsumer)***
The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if omode has its default value of "out".

[Default: -]

poll = <millisec> ***(Integer)***
Interval to wait between polling attempts, in milliseconds. Asynchronous TAP queries can only find out when they are complete by repeatedly polling the server to find out the job's status. This parameter allows you to set how often that happens. Attempts to set it too low (<50) will be rejected on the assumption that you're thinking in seconds.

[Default: 5000]

progress = true|false ***(Boolean)***
If this parameter is set true, progress of the job is reported to standard output as it happens.

[Default: true]

B.25.2 Examples

Here are some examples of tapresume:

```
stilts tapresume joburl='http://dc.zah.uni-heidelberg.de/__system__/tap/run/tap/async/d4ENGR
               out=result.csv ofmt=csv
```

Resumes waiting for the output of a query on a job with ID d4ENGR which was previously started on the GAVO TAP server. When it has completed the output table will be written as a comma-separated value file.

B.26 tapskymatch: Crossmatches table on sky position against TAP table

tapskymatch allows you to perform a positional crossmatch of a local table with one held in a remote TAP service, as long as that TAP supports upload queries. This task does three main jobs. First, it prepares the ADQL queries and TAP negotiations for you so that you don't need to remember the syntax for performing positional crossmatches against a TAP service. Second, it organises data transfer so that only those columns required (basically the positional ones) are transmitted to and from the service, to save on bandwidth. And third it divides the job up into chunks, so that the TAP service only has to perform a manageable-sized query at a time. If the job is large this chunking can be useful to monitor progress of the job, and it also allows you to perform a match which would otherwise hit the upload or output limits imposed by the service.

The positional match may be done in any spherical coordinate system, it's up to the user to ensure that the same coordinates are provided for the local and remote tables.

Note that `cdsskymatch` provides similar functionality by accessing a different external service, which is usually much faster; if the table you wish to match is part of the VizieR database, you may wish to use that command instead.

B.26.1 Usage

The usage of `tapskymatch` is

```
stilts <stilts-flags> tapskymatch ifmt=<in-format> istream=true|false
                                  icmd=<cmds> ocmd=<cmds>
                                  omode=out|meta|stats|count|checksum|cgi|discard|topcat|sa
                                  out=<out-table> ofmt=<out-format>
                                  inlon=<expr/deg> inlat=<expr/deg>
                                  tapurl=<url-value>
                                  interface=tap1.0|tap1.1|cap
                                  taptable=<name> taplon=<column>
                                  taplat=<column> tapcols=<colname,...>
                                  sr=<expr/deg>
                                  find=all|best|each|each-dist
                                  blocksize=<int-value> maxrec=<int-value>
                                  sync=true|false blockmaxrec=<nrow>
                                  compress=true|false fixcols=none|dups|all
                                  suffixin=<label> suffixremote=<label>
                                  [in=]<table>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TapUploadSkyMatch`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`blockmaxrec = <nrow>` ***(Long)***
Sets the MAXREC parameter passed to the TAP service for each uploaded block. This allows you to request that the service overrides its default limit for the number of rows output from a single query. The service may still impose some "hard" limit beyond which the output row count cannot be increased.

Note this differs from the `maxrec` parameter, which gives the maximum total number of rows to be returned from this command.

`blocksize = <int-value>` ***(Integer)***
The number of rows uploaded in each TAP query. TAP services may have limits on the number of rows in a table uploaded for matching. This command can therefore break up input tables into blocks and make a number of individual TAP queries to generate the result. This parameter controls the maximum number of rows uploaded in each individual request. For an input table with fewer rows than this value, the whole thing is done as a single query.

[Default: `5000`]

`compress = true|false` ***(Boolean)***
If true, the service is requested to provide HTTP-level compression for the response stream (Accept-Encoding header is set to "`gzip`", see RFC 2616). This does not guarantee that compression will happen but if the service honours this request it may result in a smaller amount of network traffic at the expense of more processing on the server and client.

[Default: `true`]

`find = all|best|each|each-dist` ***(UserFindMode)***
Determines which pair matches are included in the result.

- `all`: All matches

- `best`: Matched rows, best remote row for each input row
- `each`: One row per input row, contains best remote match or blank
- `each-dist`: One row per input row, column giving distance only for best match

Note only the `all` mode is symmetric between the two tables.

[Default: `all`]

`fixcols = none|dups|all` ***(Fixer)***

Determines how input columns are renamed before use in the output table. The choices are:

- `none`: columns are not renamed
- `dups`: columns which would otherwise have duplicate names in the output will be renamed to indicate which table they came from
- `all`: all columns will be renamed to indicate which table they came from

If columns are renamed, the new ones are determined by `suffix*` parameters.

[Default: `dups`]

`icmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`inlat = <expr/deg>` ***(String)***

Longitude in degrees for the position of each row in the input table. This may simply be a column name, or it may be an algebraic expression as explained in Section 10. The coordinate system must match that used for the coordinates in the remote table.

inlon = <expr/deg> ***(String)***
Longitude in degrees for the position of each row in the input table. This may simply be a column name, or it may be an algebraic expression as explained in Section 10. The coordinate system must match that used for the coordinates in the remote table.

interface = tap1.0|tap1.1|cap ***(String)***
Defines how the service endpoints and the version of the TAP protocol to use for queries is determined. This may take one of the following (case-insensitive) values:

- `TAP1.0`: The standard TAP endpoints are used, without examining the service's capabilities document. The service is queried using version 1.0 of the TAP protocol.
- `TAP1.1`: The standard TAP endpoints are used, without examining the service's capabilities document. The service is queried using version 1.1 of the TAP protocol.
- `cap`: The service's capabilities document is examined, and the endpoints listed there are used.

The capabilities document, if used, is read from `tapurl`/capabilities unless the `capabilitiesurl` parameter is defined, in which case that is used.

The baseline value of all the TAP service endpoints (`sync`, `async`, `tables`, `capabilities`, `examples`) are determined by this parameter, but each of those endpoint values may be overridden individually by other endpoint-specific parameters (`syncurl`, `asyncurl`, `tablesurl`, `capabilitiesurl`, `availabilityurl`, `examplesurl`)

For default (unauthenticated) access, the default value is usually suitable.

[Default: `tap1.0`]

istream = true|false ***(Boolean)***
If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

maxrec = <int-value> ***(Integer)***
Limit to the number of rows resulting from this operation. If the value is negative (the default) no limit is imposed. Note however that there can be truncation of the result if the number of records returned from a single chunk exceeds limits imposed by the service.

[Default: `-1`]

ocmd = <cmds> ***(ProcessingStep[])***
Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ofmt = <out-format> ***(String)***
Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special

value "(auto)" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if omode has its default value of "out".

[Default: (auto)]

omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is out, which means that the result will be written as a new table to disk or elsewhere, as determined by the out and ofmt parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (<mode-args>) are required to determine the exact behaviour.

Possible values are

- out
- meta
- stats
- count
- checksum
- cgi
- discard
- topcat
- samp
- plastic
- tosql
- gui

Use the help=omode flag or see Section 6.4 for more information.

[Default: out]

out = <out-table> ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if omode has its default value of "out".

[Default: -]

sr = <expr/deg> ***(String)***

Maximum distance in degrees from the local table (lat,lon) position at which counterparts from the remote table will be identified. This is an ADQL expression interpreted within the TAP service, so it may be a constant value or may involve columns in the remote table.

suffixin = <label> ***(String)***

If the fixcols parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from the input table.

[Default: _in]

suffixremote = <label> ***(String)***

If the fixcols parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from the TAP result table.

[Default: _tap]

sync = true|false ***(Boolean)***

Determines whether the TAP queries are submitted in synchronous or asynchronous mode. Since this command uses chunking to keep requests to a reasonable size, hopefully requests will not take too long to execute, therefore the default is synchronous (true).

[Default: `true`]

`tapcols = <colname,...>` ***(String[])***
Comma-separated list of column names to retrieve from the remote table. If no value is supplied (the default), all columns from the remote table will be returned.

`taplat = <column>` ***(String)***
Latitude in degrees for the position of each row in the remote table. This is an ADQL expression interpreted within the TAP service, typically just a column name. The coordinate system must match that used for the input table.

`taplon = <column>` ***(String)***
Longitude in degrees for the position of each row in the remote table. This is an ADQL expression interpreted within the TAP service, typically just a column name. The coordinate system must match that used for the input table.

`taptable = <name>` ***(String)***
Name of the table in the given TAP service against which the matching will be performed.

`tapurl = <url-value>` ***(URL)***
The base URL of a Table Access Protocol service. This is the bare URL without a trailing "/[a]sync".

In the usual case, the default values of the various endpoints (sync and async query submission, tables metadata, service-provided examples etc) use this URL as a parent and append standard sub-paths.

In some cases however, determination of the endpoints is more complicated, as determined by the `interface` parameter which may cause endpoints to be read from the capabilities document at `tapurl/capabilities`, and by other endpoint-specific parameters (`syncurl`, `asyncurl`, `tablesurl`, `capabilitiesurl`, `availabilityurl`, `examplesurl`) for fine tuning.

B.26.2 Examples

Here are some examples of `tapskymatch`:

```
stilts tapskymatch tapurl=http://dc.g-vo.org/tap
                   taptable=twomass.data taplon=raj2000 taplat=dej2000
                   in=dr5qso.fits inlon=RA inlat=DEC sr=0.00027 find=all
                   out=qso_2mass.fits
```

Matches a local catalogue `dr5qso.fits` against the table named `twomass.data` in the GAVO TAP service. The search radius is 1/3600 degrees (1 arcsecond) and all 2MASS sources within the radius of each input source are returned.

If you run the command with "`stilts -verbose ...`" the text of the ADQL query submitted to the TAP service will (amongst other things) be logged on the console, and you will also see the number of rows uploaded and matched in each chunk.

```
stilts tapskymatch tapurl=http://dc.g-vo.org/tap
                   taptable=rave.dr3 taplon=raj2000 taplat=dej2000
                   tapcols=name,raj2000,dej2000,pmra,pmde
                   in=hip_main.fits inlon=RAdeg inlat=DEdeg
                   icmd='keepcols "HIP RAdeg DEdeg pmra pmde"'
                   sr=0.00027
                   icmd='select nearMoc(\"III/265/ravedr3\",RAdeg,DEdeg,.00027)'
                   icmd=cache icmd=progress
                   blocksize=5000
```

```
fixcols=all suffixin=_hip suffixremote=_rave
find=best
omode=topcat
```

This matches a local copy of the Hipparcos survey against a remote copy of the RAVE survey with a 1-arcsecond radius. The output table contains only the identifier, position and proper motion columns from both the input table (by using the `keepcols` filter) and the remote table (by specifying `tapcols`); the other columns are discarded. The `fixcols` and `suffix*` parameters ensure that a suffix is added to all the output column names, `_hip` for the input (Hipparcos) columns and `_rave` for the remote (RAVE) ones.

Before uploading, the input table is preprocessed by selecting only those rows that fall within the actual footprint of the RAVE survey, by filtering with a MOC giving RAVE coverage (the RAVE dr3 MOC is also available at this URL). This step reduces the amount of data that needs to be uploaded, since only those rows in the given coverage region stand a chance of having a match in the remote table. Note use of the `nearMoc` function with the value of the match radius as the fourth parameter; this includes those objects which may be outside the actual MOC region but close enough that a match could still result.

The `blocksize` parameter determines the number of rows uploaded at a time. If you receive warnings that the output has been truncated, you should decrease this number.

Progress is displayed as the match continues. The `cache` filter must be applied upstream of (before) the `progress` filter itself for this to work, since otherwise the match processing reads all the input rows before the actual work is done, and the progress monitor completes before the match actually starts.

B.27 `tcat`: Concatenates multiple similar tables

`tcat` is a tool for concatenating any number of similar tables one after the other. The tables must be of similar form to each other (same number and types of columns). Preprocessing of the tables may be done using the `icmd` parameter, which will operate in the same way on all the input tables. Table parameters of the output table will be taken from the first of the input tables.

Subject to some constraints on the details of the input and output formats and processing, `tcat` is capable of joining an unlimited number of tables together to produce an output table of unlimited length, without large memory requirements. If there are very many input files, it may be necessary to set the `lazy` parameter so that they are not all kept open at once.

If you have heterogeneous tables, in different formats or requiring different preprocessing steps from each other before they can be concatenated, use `tcatn` instead.

B.27.1 Usage

The usage of `tcat` is

```
stilts <stilts-flags> tcat in=<table> [<table> ...] ifmt=<in-format>
                          multi=true|false istream=true|false icmd=<cmds>
                          ocmd=<cmds>
                          omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|plas
                          out=<out-table> ofmt=<out-format>
                          seqcol=<colname> loccol=<colname>
                          uloccol=<colname> lazy=true|false
                          countrows=true|false
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TableCat`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`countrows = true|false` ***(Boolean)***
Whether to count the rows in the table before starting the output. This is essentially a tuning parameter - if writing to an output format which requires the number of rows up front (such as normal FITS) it may result in skipping the number of passes through the input files required for processing. Unless you have a good understanding of the internals of the software, your best bet for working out whether to set this true or false is to try it both ways

[Default: `false`]

`icmd = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on each input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

The same format parameter applies to all the tables specified by `in`.

[Default: `(auto)`]

`in = <table> [<table> ...]` ***(TableProducer[])***
Locations of the input tables. Either specify the parameter multiple times, or supply the input tables as a space-separated list within a single use.

The following table location forms are allowed:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

Compression in any of the supported compression formats (Unix compress, gzip or bzip2) is expanded automatically.

A list of input table locations may be given in an external file by using the indirction character '@'. Thus "`in=@filename`" causes the file `filename` to be read for a list of input table locations. The locations in the file should each be on a separate line.

`istream = true|false` ***(Boolean)***
If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary

to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

The same streaming flag applies to all the tables specified by `in`.

[Default: `false`]

`lazy = true|false` ***(Boolean)***
Whether to perform table resolution lazily. If true, each table is only accessed when the time comes to add its rows to the output; if false, then all the tables are accessed up front. This is mostly a tuning parameter, and on the whole it doesn't matter much how it is set, but for joining an enormous number of tables setting it true may avoid running out of resources.

[Default: `false`]

`loccol = <colname>` ***(String)***
Name of a column to be added to the output table which will contain the location (as specified in the input parameter(s)) of the input table from which each row originated.

`multi = true|false` ***(Boolean)***
Determines whether all tables, or just the first one, from input table files will be used. If set `false`, then just the first table from each file named by `in` will be used. If `true`, then all tables present in those input files will be used. This only has an effect for file formats which are capable of containing more than one table, which effectively means FITS and VOTable and their variants.

[Default: `false`]

`ocmd = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***
Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***
The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or

populating a table in an SQL database. For some values of this parameter, additional parameters (<mode-args>) are required to determine the exact behaviour.

Possible values are

- out
- meta
- stats
- count
- checksum
- cgi
- discard
- topcat
- samp
- plastic
- tosql
- gui

Use the help=omode flag or see Section 6.4 for more information.

[Default: out]

out = <out-table> ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if omode has its default value of "out".

[Default: -]

seqcol = <colname> ***(String)***

Name of a column to be added to the output table which will contain the sequence number of the input table from which each row originated. This column will contain 1 for the rows from the first concatenated table, 2 for the second, and so on.

uloccol = <colname> ***(String)***

Name of a column to be added to the output table which will contain the unique part of the location (as specified in the input parameter(s)) of the input table from which each row originated. If not null, parameters will also be added to the output table giving the pre- and post-fix string common to all the locations. For example, if the input tables are "/data/cat_a1.fits" and "/data/cat_b2.fits" then the output table will contain a new column <colname> which takes the value "a1" for rows from the first table and "b2" for rows from the second, and new parameters "<colname>_prefix" and "<colname>_postfix" with the values "/data/cat_" and ".fits" respectively.

B.27.2 Examples

Here are some examples of tcat:

```
stilts tcat ifmt=ascii in=t1.txt in=t2.txt in=t3.txt out=table.txt
```

Concatenates the three named ASCII format tables to produce an output table. All three must have compatible numbers and types of columns.

```
stilts tcat ifmt=ascii in="t1.txt t2.txt t3.txt" out=table.txt
```

Has exactly the same effect as the previous example.

```
stilts tcat ifmt=ascii in=@inlist.lis out=table.txt
```

This will have the same effect as the previous two examples if a file name "inlist.lis" in the current directory contains three lines, "t1.txt", "t2.txt" and "t3.txt".

```
stilts tcat in=@infits.lis out=gaia_source.colfits
           lazy=true countrows=true
```

Concatenates the contents of all the files listed in "infits.lis" to one large output colfits file. The `lazy=true` is a good idea if there is a large number of input files listed.

```
stilts tcat in=r368776.fits#1 in=r368776#2 in=r368776.fits#3 in=r368776.fits#4
           out=r368776_all.fits
```

Concatenates the contents of four tables (the first four extension HDUs) from a multi-extension FITS file to produce a single FITS table. Many Unix shells (csh, bash) will allow you to list the input files using the following shorthand: "`in=r368776.fits#{1,2,3,4}`".

```
stilts tcat in=r368776.fits multi=true out=r368776_all.fits
```

Concatenates all the tables in the named file together. Setting `multi=true` means that instead of picking the first table from each named `in` table, all tables will be selected. So, if the input FITS file in this example has just four table HDUs, then this example does exactly the same as the previous one, but with less typing. The same thing works with multi-TABLE VOTable documents, but most other file formats (CSV etc) do not have the facility for storing multiple tables in a single file.

```
stilts tcat in=r368776.fits multi=true out=r368776_all.fits
           icmd=progress seqcol=ID
```

Does the same as the previous example with a couple of additions. Firstly, progress through each of the input files will be reported to the console. Secondly, an additional column "ID" will be appended to the output which contains 1 for all the rows from the first input table, 2 for the rows from the second one and so on.

```
stilts tcat in='rA.csv rB.csv rC.csv' ifmt=csv \
           icmd='keepcols "RA DEC FLUX"' icmd='sorthead 10 FLUX' \
           ocmd='sort FLUX'
```

Takes the 10 rows with highest FLUX values from each of three input tables (in comma-separated value format) and joins them together to produce a 30-row output table. This is then sorted in FLUX order, and the resulting table is output to the console in text format. Only the columns RA, DEC and FLUX are output; any other columns are discarded. The input tables don't need to have identical forms to each other, but each must have at least an RA, DEC and FLUX column.

```
stilts tcat in=vizier.xml multi=true
           icmd='keepcols "ucd$RECORD ucd$POS_EQ_RA_MAIN ucd$POS_EQ_DEC_MAIN"'
           uloccol=TID out=all.csv
```

This processes a VOTable file which may have multiple TABLEs in it, but for which each of the tables is known to have columns with the UCDs RECORD, POS_EQ_RA_MAIN and POS_EQ_DEC_MAIN (this is typical of VOTables retrieved from CDS's VizieR service). It retains only those columns from each table and writes the result as a single concatenated table to a CSV file.

B.28 `tcatn`: Concatenates multiple tables

`tcatn` is a tool for concatenating a number of tables one after the other. Each table can be

manipulated separately prior to the concatenatation. If you have two tables T1 and T2 which contain similar columns, and you want to treat them as a single table, you can use `tcatn` to produce a new table whose metadata (row headings etc) comes from T1 and whose data consists of all the rows of T1 followed by all the rows of T2.

For this concatenation to make sense, each column of T1 must be compatible with the corresponding column of T2 - they must have compatible types and, presumably, meanings. If this is not the case for the tables that you wish to concatenate, for instance the columns are in different orders, or the units differ between a column in T1 and its opposite number in T2, you can use the `icmd1` and/or `icmd2` parameters to manipulate the input tables so that the column sequences are compatible. See Appendix B.28.2 for some examples.

If the tables are similar to each other (same format, same columns, same preprocessing stages required if any), you may find it easier to use `tcat` instead.

B.28.1 Usage

The usage of `tcatn` is

```
stilts <stilts-flags> tcatn nin=<count> ifmtN=<in-format> inN=<tableN>
                            icmdN=<cmds> ocmd=<cmds>
                            omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|pla
                            out=<out-table> ofmt=<out-format>
                            seqcol=<colname> loccol=<colname>
                            uloccol=<colname> countrows=true|false
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TableCatN`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`countrows = true|false` ***(Boolean)***
Whether to count the rows in the table before starting the output. This is essentially a tuning parameter - if writing to an output format which requires the number of rows up front (such as normal FITS) it may result in skipping the number of passes through the input files required for processing. Unless you have a good understanding of the internals of the software, your best bet for working out whether to set this true or false is to try it both ways

[Default: `false`]

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on input table #N as specified by parameter `inN`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of input table #N as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the

table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <tableN>` ***(StarTable)***

The location of input table #N. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`loccol = <colname>` ***(String)***

Name of a column to be added to the output table which will contain the location (as specified in the input parameter(s)) of the input table from which each row originated.

`nin = <count>` ***(Integer)***

The number of input tables for this task. For each of the input tables N there will be associated parameters `ifmtN`, `inN` and `icmdN`.

`ocmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`
- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

`out = <out-table>` ***(TableConsumer)***
The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: -]

`seqcol = <colname>` ***(String)***
Name of a column to be added to the output table which will contain the sequence number of the input table from which each row originated. This column will contain 1 for the rows from the first concatenated table, 2 for the second, and so on.

`uloccol = <colname>` ***(String)***
Name of a column to be added to the output table which will contain the unique part of the location (as specified in the input parameter(s)) of the input table from which each row originated. If not null, parameters will also be added to the output table giving the pre- and post-fix string common to all the locations. For example, if the input tables are "/data/cat_a1.fits" and "/data/cat_b2.fits" then the output table will contain a new column <colname> which takes the value "a1" for rows from the first table and "b2" for rows from the second, and new parameters "<colname>_prefix" and "<colname>_postfix" with the values "/data/cat_" and ".fits" respectively.

B.28.2 Examples

Here are some examples of `tcatn`:

```
stilts tcatn nin=2 in1=obs1.fits in2=obs2.fits out=combined.fits
```

Concatenates two similar observation catalogues to form a combined one. In this case, both input and output tables are FITS files.

```
stilts tcatn nin=3 omode=stats in1=obs1.txt ifmt1=ascii
                               in2=obs2.xml ifmt2=votable
                               in3=obs3.fit ifmt3=fits
```

Three catalogues with similar forms but in different data formats are joined. Instead of writing the result to an output file, the resulting joined catalogue is examined to calculate its statistics, which are written to standard output.

```
stilts tcatn nin=2 in1=survey.vot.gz ifmt2=csv in2=more_data.csv
             icmd1='addskycoords fk5 galactic RA2000 DEC2000 GLON GLAT' \
             icmd1='keepcols "OBJ_ID GLON GLAT"' \
             icmd2='keepcols "ident gal_long gal_lat"' \
             loccol=FILENAME
             omode=topcat
```

In this case we are trying to concatenate results from two tables which are quite dissimilar to each other. In the first place, one is a VOTable (no `ifmt1` parameter is required since VOTables can be detected automatically), and the other is a comma-separated-values file (for which the `ifmt2=csv` parameter must be given). In the second place, the column structure of the two tables may be quite different. By pre-processing the two tables using the `icmd1` & `icmd2` parameters, we produce in each case an input table which consists of three columns of compatible types and meanings: an integer identifier and floating point galactic longitude and latitude coordinates. The second table contains such columns to start with, but the first table requires an initial step to convert FK5 J2000.0 coordinates to galactic ones. `tcatn` joins the two doctored tables together, to produce a table which contains only these three columns, with all the rows from both input tables, and sends the result directly to a new or running instance of TOPCAT. An additional column named FILENAME is appended to the table before sending it; this contains "survey.vot.gz" for all the columns from the first table and "more_data.csv" for all the columns from the second one.

B.29 `tcopy`: Converts between table formats

`tcopy` is a table copying tool. It simply copies a table from one place to another, but since you can specify the input and output formats as desired, it works as a converter from any of the supported input formats (Section 5.1.1) to any of the supported output formats (Section 5.1.2).

`tcopy` is just a stripped-down version of `tpipe` - it doesn't do anything that `tpipe` can't, but the usage is slightly simplified. It is provided as a drop-in replacement for the old `tablecopy` (`uk.ac.starlink.table.TableCopy`) tool which was supplied with earlier versions of STIL and TOPCAT - it has the same arguments and behaviour as `tablecopy`, but is implemented somewhat differently and will in some cases be more efficient.

B.29.1 Usage

The usage of `tcopy` is

```
stilts <stilts-flags> tcopy ifmt=<in-format> ofmt=<out-format>
                            [in=]<table> [out=]<out-table>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TableCopy`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`ifmt = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

in = <table> ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "-", meaning standard input. In this case the input format must be given explicitly using the ifmt parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form :<scheme-name>:<scheme-args>.
- A system command line with either a "<" character at the start, or a "|" character at the end ("<syscmd" or "syscmd|"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

ofmt = <out-format> ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "(auto)" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

[Default: (auto)]

out = <out-table> ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

[Default: -]

B.29.2 Examples

Here are some examples of tcopy in use:

stilts tcopy stars.fits stars.xml

Copies a FITS table to a VOTable. Since no input format is specified, the format is automatically detected (FITS is one of the formats for which this is possible). Since no output format is specified, the stars.xml filename is examined to make a guess at the kind of output to write: the .xml ending is taken to mean a TABLEDATA-encoded VOTable.

stilts tcopy stars.fits stars.xml ifmt=fits ofmt=votable

Does the same as the previous example, but the input and output formats have been specified explicitly.

stilts tcopy ofmt=text http://remote.host/data/vizer.xml.gz#4 -

Prints the contents of a remote, compressed VOTable to the terminal in a human-readable form. The #4 at the end of the URL indicates that the data from the fifth TABLE element in the remote document are to be used. The gzip compression of the table is taken care of automatically.

stilts tcopy ifmt=csv ofmt=latex spec.csv

Converts a comma-separated values file to a LaTeX table environment, writing the result to standard output.

```
stilts -classpath /usr/local/jars/pg73jdbc3.jar \
       -Djdbc.drivers=org.postgresql.Driver \
       tcopy in="jdbc:postgresql://localhost/imsim#SELECT ra, dec, Imag FROM dqc" \
             ofmt=fits wfslist.cat
```

Makes an SQL query on a PostgreSQL database and writes the results to a FITS file. The whole command is shown here, to show that the classpath is augmented to include the PostgreSQL driver class, and the driver class is named using the `jdbc.drivers` system property. As you can see, using SQL from Java is a bit fiddly, and there are other ways to perform this setup than on the command line - see Section 3.4 and `tpipe`'s `omode=tosql` output mode.

B.30 `tcube`: Calculates N-dimensional histograms

`tcube` constructs an N-dimensional histogram, or density map, from N columns of an input table, and writes it out as an N-dimensional data cube. The parameters you supply define which N numeric columns of the input table you want to use and the dimensions (bounds and pixel sizes) of the output grid, as well as any weighting to be applied to each point and how the weighted quantities in a single bin are to be aggregated together. Each table row then defines a point in N-dimensional space. The program goes through each row, and if the point that row defines falls within the bounds of the output grid you have defined, associates the weight value with the relevant pixel in the output grid. When all the input values have been processed, the weights in each pixel are aggregated according to the requested combination method.

The resulting N-dimensional array, whose pixel values represent an aggregation of the rows associated with that region of the N-dimensional space, is then written out as a FITS file. In one dimension, this gives you a normal histogram of a given variable. In two dimensions it might typically be used to plot the density or weighted density on the sky of objects from a catalogue.

As with some of the other generic table commands, you can perform extensive pre-processing on the input table by use of the `icmd` parameter before the actual cube counts are calculated.

See also `tgridmap`, which does a similar job to this command but writes the output in table format.

B.30.1 Usage

The usage of `tcube` is

```
stilts <stilts-flags> tcube cols=<expr> ... ... ifmt=<in-format>
                            istream=true|false icmd=<cmds>
                            bounds=[<lo>]:[<hi>] ... binsizes=<size> ...
                            nbins=<num> ...
                            combine=sum|sum-per-unit|count|count-per-unit|mean|median|Q1|Q3
                            out=<out-file>
                            otype=byte|short|int|long|float|double
                            scale=<expr>
                            [in=]<table>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TableCube`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`binsizes = <size> ...` ***(Double[])***

Gives the extent of of the data bins (cube pixels) in each dimension in data coordinates. The form of the value is a space-separated list of values, giving a list of extents for the first, second,

... dimension. Either this parameter or the `nbins` parameter must be supplied.

`bounds = [<lo>]:[<hi>] ...` ***(double[][])***

Gives the bounds for each dimension of the cube in data coordinates. The form of the value is a space-separated list of words, each giving an optional lower bound, then a colon, then an optional upper bound, for instance "1:100 0:20" to represent a range for two-dimensional output between 1 and 100 of the first coordinate (table column) and between 0 and 20 for the second. Either or both numbers may be omitted to indicate that the bounds should be determined automatically by assessing the range of the data in the table. A null value for the parameter indicates that all bounds should be determined automatically for all the dimensions.

If any of the bounds need to be determined automatically in this way, two passes through the data will be required, the first to determine bounds and the second to populate the cube.

`cols = <expr> ... ...` ***(String[])***

Columns to use for this task. One or more `<expr>` elements, separated by spaces, should be given. Each one represents a numeric value from the table, provided as a column name or algebraic expression.

The number of columns listed in the value of this parameter defines the dimensionality of the output data cube.

`combine =`
`sum|sum-per-unit|count|count-per-unit|mean|median|Q1|Q3|min|max|stdev|hit`
(Combiner)

Defines how values contributing to the same density map bin are combined together to produce the value assigned to that bin. Possible values are:

- `sum`: the sum of all the combined values per bin
- `sum-per-unit`: the sum of all the combined values per unit of bin size
- `count`: the number of non-blank values per bin (weight is ignored)
- `count-per-unit`: the number of non-blank values per unit of bin size (weight is ignored)
- `mean`: the mean of the combined values
- `median`: the median
- `Q1`: first quartile
- `Q3`: third quartile
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `stdev`: the sample standard deviation of the combined values
- `hit`: 1 if any values present, NaN otherwise (weight is ignored)

[Default: `sum`]

`icmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the

table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istream = true|false` ***(Boolean)***

If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`nbins = <num> ...` ***(Integer[])***

Gives the number of bins (cube pixels) in each dimension. The form of the value is a space-separated list of integers, giving the number of pixels for the output cube in the first, second, ... dimension. Either this parameter or the `binsizes` parameter must be supplied.

`otype = byte|short|int|long|float|double` ***(Class)***

The type of numeric value which will fill the output array. If no selection is made, the output type will be determined automatically as the shortest type required to hold all the values in the array. Currently, integers are always signed (no BSCALE/BZERO), so for instance the largest value that can be recorded in 8 bits is 127.

`out = <out-file>` ***(uk.ac.starlink.util.Destination)***

The location of the output file. This is usually a filename to write to. If it is equal to the special value "-" the output will be written to standard output.

The output cube is currently written as a single-HDU FITS file.

[Default: -]

`scale = <expr>` ***(String)***

Optionally gives a weight for each entry contributing to histogram bins. The value of this expression is accumulated, in accordance with the `combine` parameter, into the bin defined by its coordinates. If no expression is given, the value 1 is assumed.

B.30.2 Examples

```
stilts tcube in=2QZ_6QZ_pubcat.fits out=ccm.fits \
```

```
            cols='Bj_R U_Bj Bj' binsizes='0.05 0.05 0.5' bounds='-2:1 -3:2 :'
```

Calculates a 3-dimensional colour-colour-magnitude grid from three existing columns in a table. The bin (pixel) sizes are specified. The data bounds are specified explicitly for the (first two) colour dimensions, but for the (third) magnitude dimension it is determined from the minimum and maximum values the data in that column of the table. The output is a three-dimensional FITS cube containing the number of points in each cell.

```
stilts tcube in=iras_psc.vot out=flux60_map.fits \
           icmd='addskycoords fk5 galactic ra dec glat glon' \
           cols='glat glon' nbins='400 200' \
           scale=Fnu_60 combine=sum
```

Calculates a map of integrated fluxes in galactic coordinates from a catalogue of IRAS point sources. The output is a two-dimensional FITS image representing the sky in galactic coordinates, with each pixel containing the sum of `Fnu_60` fluxes from objects in that rectangle of `(glat,glon>` space. Bounds are determined automatically from the data, and the number of pixels in each dimension (400 in latitude and 200 in longitude) are specified, which means that the pixel sizes don't have to be. Since the input table contains sky positions in equatorial coordinates rather than galactic ones, the `addskycoords` filter is used to preprocess the data before the cube generation step (see Section 6.1).

B.31 `tloop`: Generates a single-column table from a loop variable

`tloop` generates a one-column table where the values in the column are effectively populated from a for loop (start, end, step). This may be useful as it is, or it can be postprocessed with `ocmd` parameters to add more columns etc.

B.31.1 Usage

The usage of `tloop` is

```
 stilts <stilts-flags> tloop ocmd=<cmds>
                             omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|pla
                             out=<out-table> ofmt=<out-format>
                             forcefloat=true|false
                             [colname=]<value> [start=]<float-value>
                             [end=]<float-value> [step=]<float-value>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TableLoop`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`colname = <value>` ***(String)***
Gives the name of the single column produced by this command.

[Default: `i`]

`end = <float-value>` ***(Double)***
Gives the value which the loop variable will not exceed. Exceeding is in the positive or negative sense according to the sense of the `step` parameter, as usual for a `for`-type loop.

`forcefloat = true|false` ***(Boolean)***
Affects the data type of the loop variable column. If true, the column is always floating point. If false, and if the other parameters are all of integer type, the column will be an integer column.

[Default: false]

ocmd = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "@filename" causes the file filename to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '#' character are ignored.

ofmt = <out-format> ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "(auto)" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if omode has its default value of "out".

[Default: (auto)]

omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is out, which means that the result will be written as a new table to disk or elsewhere, as determined by the out and ofmt parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (<mode-args>) are required to determine the exact behaviour.

Possible values are

- out
- meta
- stats
- count
- checksum
- cgi
- discard
- topcat
- samp
- plastic
- tosql
- gui

Use the help=omode flag or see Section 6.4 for more information.

[Default: out]

out = <out-table> ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if omode has its default value of "out".

[Default: -]

start = <float-value> ***(Double)***

Gives the starting value of the loop variable. This will the the value in the first row of the table.

[Default: `0.0`]

`step = <float-value>` ***(Double)***

Amount by which the loop variable will be incremented at each iteration, i.e. each table row.

[Default: `1.0`]

B.31.2 Examples

```
stilts tloop COUNTER 0 1000
```

Generates a table with a single column, named `COUNTER`, and a thousand rows. The value in the first row is 0 and in the last row is 999. The table is written to standard output.

```
stilts tloop time 0 10 0.25 out=times.csv
```

Generates a table with one column `time` counting from 0 to 9.75 in steps of 0.25. Output is to a CSV file. The parameters here are specified in order, but could equivalently be given by name: "`stilts tloop var=time start=0 end=10 step=0.26`".

```
stilts tloop x start=1 end=11 ocmd='addcol x2 x*x' ocmd='addcol x3 x*x*x'
           ocmd='stats name sum'
```

Generates a table with a column `x` running from 1 to 10 inclusive. The `addcol` filters then append two further columns, giving the squares and cubes of these values respectively, giving a table of 10 rows and 3 columns. Finally this table is piped through a `stats` filter to calculate the sums of the values, squares and cubes in this range.

B.32 `tgridmap`: Calculates N-dimensional density maps

`tgridmap` scans an input table to create one or more N-dimensional density maps, or equivalently N-dimensional histograms, of the values in an input table, and outputs the result as an, optionally sparse, table containing a row for each grid cell. The maps/histograms can optionally be weighted by some quantity from the input table, and various options such as summing, averaging and counting are available for aggregation of inputs into the output bins.

The supplied `coords` parameter defines which N numeric columns of the input table form the coordinates of the bin grid, and the `cols` parameter defines which quantities are aggregated into each bin. Either the `binsizes` or `nbins` parameter must be supplied to define the extents of the bins on each axis. The output table contains a row for each bin, with columns giving the central (and upper/lower bound) values of each grid coordinate, and a column for each aggregated value. The rows are output in first-coordinate-slowest sequence, and the `sparse` parameter determines whether a row is written for every cell in the hypercube defined by the grid dimensions, or only for those cells with non-blank data.

The tabular form of the output may not be the most appropriate or compact way to write a density map, especially for multi-dimensional grids, but it means the output can be manipulated later by other STILTS commands or by TOPCAT. To do a similar job with more compact output, see `tcube`. See also `tskymap`, which does the same thing for sky geometry (and is probably a better choice if you find yourself accumulating onto a longitude-latitude grid).

B.32.1 Usage

The usage of `tgridmap` is

```
   stilts <stilts-flags> tgridmap ifmt=<in-format> istream=true|false
                                  icmd=<cmds> ocmd=<cmds>
                                  omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|
                                  out=<out-table> ofmt=<out-format>
                                  coords=<expr> ... logs=true|false ...
                                  bounds=[<lo>]:[<hi>] ... binsizes=<size> ...
                                  nbins=<num> ...
                                  cols=<expr>[;<combiner>[;<name>]] ...
                                  combine=sum|sum-per-unit|count|count-per-unit|mean|median|Q1
                                  sparse=true|false
                                  [in=]<table>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.GridDensityMap`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`binsizes = <size> ...` ***(Double[])***

Gives the extent of of the data bins in each dimension in data coordinates. The form of the value is a space-separated list of values, giving a list of extents for the first, second, ... dimension. Either this parameter or the `nbins` parameter must be supplied.

If supplied, this parameter must have the same number of words as the `coords` parameter.

`bounds = [<lo>]:[<hi>] ...` ***(double[][])***

Gives the bounds for each dimension of the cube in data coordinates. The form of the value is a space-separated list of words, each giving an optional lower bound, then a colon, then an optional upper bound, for instance "1:100 0:20" to represent a range for two-dimensional output between 1 and 100 of the first coordinate (table column) and between 0 and 20 for the second. Either or both numbers may be omitted to indicate that the bounds should be determined automatically by assessing the range of the data in the table. A null value for the parameter indicates that all bounds should be determined automatically for all the dimensions.

If any of the bounds need to be determined automatically in this way, two passes through the data will be required, the first to determine bounds and the second to calculate the map.

If supplied, this parameter must have the same number of words as the `coords` parameter.

`cols = <expr>[;<combiner>[;<name>]] ...` ***(String[])***

Defines the quantities to be calculated. The value is a space-separated list of items, one for each aggregated column in the output table.

Each item is composed of one, two or three tokens, separated by semicolon (";") characters:

- `<expr>`: *(required)* column name or expression using the expression language for the quantity to be aggregated.
- `<combiner>`: *(optional)* combination method, using the same options as for the `combine` parameter. If omitted, the value specified for that parameter will be used.
- `<name>`: *(optional)* name of output column; if omitted, the `<expr>` value (perhaps somewhat sanitised) will be used.

It is often sufficient just to supply a space-separated list of input table column names for this parameter, but the additional syntax may be required for instance if it's required to calculate both a sum and mean of the same input column.

The default value is "`1;count;COUNT`" which simply provides an unweighted histogram, i.e. a count of the rows in each bin (aggregation of the value "`1`" using the combination method "`count`", yielding an output column named "`COUNT`").

[Default: `1;count;COUNT`]

`combine =`
`sum|sum-per-unit|count|count-per-unit|mean|median|Q1|Q3|min|max|stdev|hit`
(Combiner)

Defines the default way that values contributing to the same density map bin are combined together to produce the value assigned to that bin. Possible values are:

- `sum`: the sum of all the combined values per bin
- `sum-per-unit`: the sum of all the combined values per unit of bin size
- `count`: the number of non-blank values per bin (weight is ignored)
- `count-per-unit`: the number of non-blank values per unit of bin size (weight is ignored)
- `mean`: the mean of the combined values
- `median`: the median
- `Q1`: first quartile
- `Q3`: third quartile
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `stdev`: the sample standard deviation of the combined values
- `hit`: 1 if any values present, NaN otherwise (weight is ignored)

Note this value may be overridden on a per-column basis by the `cols` parameter.

[Default: `mean`]

`coords = <expr> ...` ***(String[])***

Defines the dimensions of the grid over which accumulation will take place. The form of this value is a space-separated list of words each giving a column name or algebraic expression defining one of the dimensions of the output grid. For a 1-dimensional histogram, only one value is required.

`icmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.

- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istream = true|false` ***(Boolean)***

If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`logs = true|false ...` ***(Boolean[])***

Determines whether each coordinate axis is linear or logarithmic. By default the grid axes are linear, but if this parameter is supplied with one or more true values, the bins on the corresponding axes are assigned logarithmically instead.

If supplied, this parameter must have the same number of words as the `coords` parameter.

`nbins = <num> ...` ***(Integer[])***

Gives the approximate number of bins in each dimension. The form of the value is a space-separated list of integers, giving the number of bins for the output histogram in the first, second, ... dimension. An attempt is made to use round numbers for bin sizes so the bin counts may not be exactly as specified. Either this parameter or the `binsizes` parameter must be supplied.

If supplied, this parameter must have the same number of words as the `coords` parameter.

`ocmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that

the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`
- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

`out = <out-table>` ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `-`]

`sparse = true|false` ***(Boolean)***

Determines whether a row is written for every cell in the defined grid, or only for those cells in which data appears in the input. The result will usually be more compact if this is set false, but if you want to compare results from different runs it may be convenient to set it true.

[Default: `true`]

B.32.2 Examples

Here are some examples of using `tgridmap`:

```
stilts tgridmap in=ravedr4.fits coords=HRV nbins=20
```

Calculates a simple 1-dimensional unweighted histogram of the HRV column from the input table `ravedr4.fits`. The output is a table with columns `HRV`, giving the central value of each bin, and `COUNT`, giving the number of input rows with HRV values in that bin; additional columns `HRV_lo` and `HRV_hi` give the lower and upper bounds of the bin. The bin size is determined from the actual range of the HRV values in the input table, combined with the requested bin count of 20; however, the bin size will be chosen as some round number, so the bin count (number of rows in the output table) may not be exactly as requested.

```
stilts tgridmap in=ravedr4.fits coords=HRV
               binsizes=100 bounds=-450:450 sparse=false
```

Produces a similar histogram to the previous example, but the bin dimensions, ranges and alignments are specified explicitly rather than being worked out from the data. There will be 9 output bins, [-450,-350), [-350,-250), ..., [350,450); any values outside of these bins will be

ignored. The `sparse=false` parameter means that rows will be output for all 9 bins, even if some of them are empty. Note supplying bin geometry in this way allows control of bin boundaries; in this case HRV=0 is in the middle of a bin not at a bin boundary. This will also be faster, since no initial scan to determine actual data ranges has to be performed.

```
stilts tgridmap in=edr3-local.fits
               icmd='addcol nobs astrometric_n_good_obs_al'
               icmd='addcol g_abs phot_g_mean_mag+5*log10(parallax*0.01)'
               coords='bp_rp g_abs'
               binsizes='0.125 0.5' bounds='-1:6 -5:20'
               cols='1;count;NUM nobs;sum;SUM_NOBS nobs;mean;MEAN_NOBS'
               out=grid-stats.vot sparse=false
```

This assembles a table containing three weighted histograms on a 2-d colour vs. absolute-magnitude grid. The output table contains columns giving `bp_rp` and `g_abs` coordinate values for each grid point, as well as columns NUM containing source density, and columns SUM_NOBS and MEAN_NOBS containing respectively the sum and mean of the `nobs` column in each grid cell. Since `sparse=false` the number and arrangement of output rows is determined by the binsizes and bounds (57*51 rows) independent of the input data, and could be compared with similar runs on different input tables. The `icmd=addcol...` parameters prepare values for accumulation ahead of the actual gridding step for convenience though this isn't essential, the relevant expressions could be used directly in the `coords` and `cols` parameters if preferred.

B.33 `tjoin`: Joins multiple tables side-to-side

`tjoin` performs a trivial side-by-side join of multiple tables. The N'th row of the output table consists of the N'th row of the first input table, followed by the N'th row of the second input table, ... and so on. It is suitable if you want to amalgamate two or more tables whose row orderings correspond exactly to each other.

For the (more usual) case in which the rows of the tables to be joined are not already in the right order, use one of the crossmatching commands (Section 7).

B.33.1 Usage

The usage of `tjoin` is

```
stilts <stilts-flags> tjoin nin=<count> ifmtN=<in-format> inN=<tableN>
                           icmdN=<cmds> ocmd=<cmds>
                           omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|pla
                           out=<out-table> ofmt=<out-format>
                           fixcols=none|dups|all suffixN=<label>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TableJoinN`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`fixcols = none|dups|all` ***(Fixer)***

Determines how input columns are renamed before use in the output table. The choices are:

- `none`: columns are not renamed
- `dups`: columns which would otherwise have duplicate names in the output will be renamed to indicate which table they came from
- `all`: all columns will be renamed to indicate which table they came from

If columns are renamed, the new ones are determined by `suffix*` parameters.

[Default: `dups`]

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on input table #N as specified by parameter `inN`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of input table #N as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <tableN>` ***(StarTable)***
The location of input table #N. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`nin = <count>` ***(Integer)***
The number of input tables for this task. For each of the input tables N there will be associated parameters `ifmtN`, `inN` and `icmdN`.

`ocmd = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***
Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 -

matching is case-insensitive and you can use just the first few letters). If it has the special value "(auto)" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if omode has its default value of "out".

[Default: (auto)]

omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is out, which means that the result will be written as a new table to disk or elsewhere, as determined by the out and ofmt parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (<mode-args>) are required to determine the exact behaviour.

Possible values are

- out
- meta
- stats
- count
- checksum
- cgi
- discard
- topcat
- samp
- plastic
- tosql
- gui

Use the help=omode flag or see Section 6.4 for more information.

[Default: out]

out = <out-table> ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if omode has its default value of "out".

[Default: -]

suffixN = <label> ***(String)***

If the fixcols parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from table N.

[Default: _N]

B.33.2 Examples

Here are some examples of using tjoin:

stilts tjoin nin=2 in1=positions.fit in2=fluxes.fits out=combined.fits

Takes two input FITS files and sticks them together side by side, writing the result as a third FITS file. The output will have the same number of rows as each of the input catalogues, and a number of columns equal to the sum of those in the two input catalogues.

```
stilts tjoin nin=3 fixcols=all \
             ifmt1=ascii in1=t1.txt suffix1=_T1 \
             ifmt2=ascii in2=t2.txt suffix2=_T2 \
             ifmt3=ascii in3=t3.txt suffix3=_T3 \
             ocmd='select FLAG_T1==0' \
             omode=stats
```

This joins three ascii tables together. Each column of the output table is renamed by appending a string to it ("_T1" for the first table, "_T2" for the second...). Only those rows of the output for which the FLAG column in the first input table, and hence the FLAG_T1 column in the output table, have the value zero are selected. Statistics are calculated for all the columns of these selected rows, and written to the output.

B.34 `tmatch1`: Performs a crossmatch internal to a single table

`tmatch1` performs efficient and flexible crossmatching between the rows of a single table. It can match rows on the basis of their relative position in the sky, or alternatively using many other criteria such as separation in in some isotropic or anisotropic Cartesian space, identity of a key value, or some combination of these; the full range of match criteria is dicussed in Section 7.1.

The basic task performed by the intra-table matcher is to identify groups of rows within the table which match each other. See Section 7.2 for an explanation of exactly what consitutes a match group. The result of identifying these groups is expressed as an output table in one of a variety of ways, specified by the `action` parameter. These options include marking group membership in added columns and eliminating some or all rows which form part of a match group.

B.34.1 Usage

The usage of `tmatch1` is

```
stilts <stilts-flags> tmatch1 matcher=<matcher-name> params=<match-params>
                              tuning=<tuning-params> values=<expr-list>
                              action=identify|keep0|keep1|wide2|wideN
                              progress=none|log|profile
                              runner=classic|parallel|sequential|partest
                              ifmt=<in-format> istream=true|false
                              icmd=<cmds> ocmd=<cmds>
                              omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|p
                              out=<out-table> ofmt=<out-format>
                              [in=]<table>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TableMatch1`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`action = identify|keep0|keep1|wide2|wideN` ***(Match1Type)***

Determines the form of the table which will be output as a result of the internal match.

- `identify`: The output table is the same as the input table except that it contains two additional columns, `GroupID` and `GroupSize`, following the input columns. Each group of rows which matched is assigned a unique integer, recorded in the GroupID column, and the size of each group is recorded in the GroupSize column. Rows which don't match any others (singles) have null values in both these columns.
- `keep0`: The result is a new table containing only "single" rows, that is ones which don't match any other rows in the table. Any other rows are thrown out.

- `keep1`: The result is a new table in which only one row (the first in the input table order) from each group of matching ones is retained. A subsequent intra-table match with the same criteria would therefore show no matches.
- `wideN`: The result is a new "wide" table consisting of matched rows in the input table stacked next to each other. Only groups of exactly N rows in the input table are used to form the output table; each row of the output table consists of the columns of the first group member, followed by the columns of the second group member and so on. The output table therefore has N times as many columns as the input table. The column names in the new table have `_1`, `_2`, ... appended to them to avoid duplication.

[Default: `identify`]

`icmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istream = true|false` ***(Boolean)***

If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

matcher = <matcher-name> ***(MatchEngine)***

Defines the nature of the matching that will be performed. Depending on the name supplied, this may be positional matching using celestial or Cartesian coordinates, exact matching on the value of a string column, or other things. A list and explanation of the available matching algorithms is given in Section 7.1. The value supplied for this parameter determines the meanings of the values required by the `params`, `values*` and `tuning` parameter(s).

[Default: `sky`]

ocmd = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ofmt = <out-format> ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`
- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

out = <out-table> ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: -]

`params = <match-params>` ***(String[])***

Determines the parameters of this match. This is typically one or more tolerances such as error radii. It may contain zero or more values; the values that are required depend on the match type selected by the `matcher` parameter. If it contains multiple values, they must be separated by spaces; values which contain a space can be 'quoted' or "quoted".

`progress = none|log|profile` ***(String)***

Determines whether information on progress of the match should be output to the standard error stream as it progresses. For lengthy matches this is a useful reassurance and can give guidance about how much longer it will take. It can also be useful as a performance diagnostic.

The options are:

- `none`: no progress is shown
- `log`: progress information is shown
- `profile`: progress information and limited time/memory profiling information are shown

[Default: `log`]

`runner = classic|parallel|sequential|partest` ***(RowRunner)***

Selects the threading implementation. The options are currently:

- `classic`: uses legacy sequential implementation
- `parallel`: uses multithreaded implementation for large tables
- `sequential`: uses multithreaded implementation but with only a single thread
- `partest`: uses multithreaded implementation even when tables are small

The `parallel` option should run faster than the others, at least for large matches and where multiple processing cores are available. At present however, this code is less well tested than the legacy implementation, so `classic` is the default. `sequential` and `partest` are mostly provided for testing purposes.

`parallel` ought to run correctly, but if you want to be sure of the results, you can start by comparing the results from `sequential` and `parallel` and then use `parallel` for larger matches.

Changing this parameter should make no difference to the matching results. If you notice any discrepancies **please report them**.

[Default: `classic`]

`tuning = <tuning-params>` ***(String[])***

Tuning values for the matching process, if appropriate. It may contain zero or more values; the values that are permitted depend on the match type selected by the `matcher` parameter. If it contains multiple values, they must be separated by spaces; values which contain a space can be 'quoted' or "quoted". If this optional parameter is not supplied, sensible defaults will be chosen.

`values = <expr-list>` ***(String[])***

Defines the values from the input table which are used to determine whether a match has occurred. These will typically be coordinate values such as RA and Dec and perhaps some per-row error values as well, though exactly what values are required is determined by the kind of match as determined by `matcher`. Depending on the kind of match, the number and type of the values required will be different. Multiple values should be separated by whitespace; if whitespace occurs within a single value it must be 'quoted' or "quoted". Elements of the expression list are commonly just column names, but may be algebraic expressions calculated

from zero or more columns as explained in Section 10.

B.34.2 Examples

Here are some examples of using `tmatch1`:

```
stilts tmatch1 matcher=sky values="RA2000 DE2000" params=20 \
               action=keep0 in=crowded.vot out=sparse.vot
```

Copies an input catalogue "crowded.vot" to an output catalogue "sparse.vot", but omitting any objects (rows) which are within 20 arcsec of other objects. The output catalogue will contain no near neighbours.

```
stilts tmatch1 matcher=skyerr values="RA2000 DE2000 RADIUS*4" params=40 \
               action=keep0 in=crowded.vot out=sparse.vot
```

This is similar to the previous example, but uses the `skyerr` matcher which determines the proximity threshold on a row-by-row basis from values in the table - in this case 4 times the value of the RADIUS column (this value must be in arc seconds). The `params=40` value does not affect the result, but it gives the algorithm an idea of the rough scale of object separation.

```
stilts tmatch1 matcher=3d values="XPIX YPIX ZPIX" params=10 action=identify \
               in=state.fit ocmd='select GroupSize>3' out=groups3+.fit
```

Uses the "3d" matcher to identify groups of objects in terms of their proximity in a 3-dimensional Cartesian space, with positions given by the XPIX, YPIX and ZPIX columns in the input table. The `action=identify` parameter means that the input table is written out with the same rows, but with additional columns indicating which rows are associated with each other. One of these columns, "GroupSize" gives the number of objects in each group. The postprocessing filter `ocmd='select GroupSize>3'` selects only those rows which are part of groups of three objects or larger; singletons and pairs are discarded before writing the output file.

```
stilts tmatch1 matcher=sky values="ra dec" params=3 action=wide2 \
               ocmd='keepcols "id_1 ra_1 dec_1 id_2 ra_2 dec_2"'
               in=galaxy.fits out=binaries.txt ofmt=ascii
```

Identifies pairs of objects within 3 arcsec of each other from an input catalogue. Singles, and groups of three or more, will be discarded. The output table generated is a double-width version of the input table with pairs of objects next to each other on the same row. Here, the `ocmd` post-processing filter discards all of the columns except the identifiers and sky positions for each object. The output is to a text file.

B.35 `tmatch2`: Crossmatches 2 tables using flexible criteria

`tmatch2` is an efficient and highly configurable tool for crossmatching pairs of tables. It can match rows between tables on the basis of their relative position in the sky, or alternatively using many other criteria such as separation in some isotropic or anisotropic Cartesian space, identity of a key value, or some combination of these; the full range of match criteria is discussed in Section 7.1. You can choose whether you want to identify all the matches or only the closest, and what form the output table takes, for instance matched rows only, or all rows from one or both tables, or only the unmatched rows.

If you simply want to match two tables based on sky position with a fixed maximum separation, you may find the `tskymatch2` command easier to use.

Note: the `duptag1` and `duptag2` parameters have been replaced at version 1.4 by `suffix1` and `suffix2` for consistency with other table join tasks.

B.35.1 Usage

The usage of `tmatch2` is

```
stilts <stilts-flags> tmatch2 ifmt1=<in-format> ifmt2=<in-format>
                              icmd1=<cmds> icmd2=<cmds> ocmd=<cmds>
                              omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|p
                              out=<out-table> ofmt=<out-format>
                              matcher=<matcher-name> values1=<expr-list>
                              values2=<expr-list> params=<match-params>
                              tuning=<tuning-params>
                              join=1and2|1or2|all1|all2|1not2|2not1|1xor2
                              find=all|best|best1|best2
                              fixcols=none|dups|all suffix1=<label>
                              suffix2=<label> scorecol=<col-name>
                              progress=none|log|profile
                              runner=classic|parallel|sequential|partest
                              [in1=]<table1> [in2=]<table2>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TableMatch2`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`find = all|best|best1|best2` ***(PairMode)***

Determines what happens when a row in one table can be matched by more than one row in the other table. The options are:

- `all`: All matches. Every match between the two tables is included in the result. Rows from both of the input tables may appear multiple times in the result.
- `best`: Best match, symmetric. The best pairs are selected in a way which treats the two tables symmetrically. Any input row which appears in one result pair is disqualified from appearing in any other result pair, so each row from both input tables will appear in at most one row in the result.
- `best1`: Best match for each Table 1 row. For each row in table 1, only the best match from table 2 will appear in the result. Each row from table 1 will appear a maximum of once in the result, but rows from table 2 may appear multiple times.
- `best2`: Best match for each Table 2 row. For each row in table 2, only the best match from table 1 will appear in the result. Each row from table 2 will appear a maximum of once in the result, but rows from table 1 may appear multiple times.

The differences between `best`, `best1` and `best2` are a bit subtle. In cases where it's obvious which object in each table is the best match for which object in the other, choosing betwen these options will not affect the result. However, in crowded fields (where the distance between objects within one or both tables is typically similar to or smaller than the specified match radius) it will make a difference. In this case one of the asymmetric options (`best1` or `best2`) is usually more appropriate than `best`, but you'll have to think about which of them suits your requirements. The performance (time and memory usage) of the match may also differ between these options, especially if one table is much bigger than the other.

[Default: `best`]

`fixcols = none|dups|all` ***(Fixer)***

Determines how input columns are renamed before use in the output table. The choices are:

- `none`: columns are not renamed

- `dups`: columns which would otherwise have duplicate names in the output will be renamed to indicate which table they came from
- `all`: all columns will be renamed to indicate which table they came from

If columns are renamed, the new ones are determined by `suffix*` parameters.

[Default: `dups`]

`icmd1 = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the first input table as specified by parameter `in1`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`icmd2 = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on the second input table as specified by parameter `in2`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt1 = <in-format>` ***(String)***
Specifies the format of the first input table as specified by parameter `in1`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`ifmt2 = <in-format>` ***(String)***
Specifies the format of the second input table as specified by parameter `in2`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in1 = <table1>` ***(StarTable)***
The location of the first input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt1` parameter. Note that not all formats can be streamed in this way.

- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`in2 = <table2>` ***(StarTable)***

The location of the second input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt2` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`join = 1and2|1or2|all1|all2|1not2|2not1|1xor2` ***(JoinType)***

Determines which rows are included in the output table. The matching algorithm determines which of the rows from the first table correspond to which rows from the second. This parameter determines what to do with that information. Perhaps the most obvious thing is to write out a table containing only rows which correspond to a row in both of the two input tables. However, you may also want to see the unmatched rows from one or both input tables, or rows present in one table but unmatched in the other, or other possibilities. The options are:

- `1and2`: An output row for each row represented in both input tables (INNER JOIN)
- `1or2`: An output row for each row represented in either or both of the input tables (FULL OUTER JOIN)
- `all1`: An output row for each matched or unmatched row in table 1 (LEFT OUTER JOIN)
- `all2`: An output row for each matched or unmatched row in table 2 (RIGHT OUTER JOIN)
- `1not2`: An output row only for rows which appear in the first table but are not matched in the second table
- `2not1`: An output row only for rows which appear in the second table but are not matched in the first table
- `1xor2`: An output row only for rows represented in one of the input tables but not the other one

[Default: `1and2`]

`matcher = <matcher-name>` ***(MatchEngine)***

Defines the nature of the matching that will be performed. Depending on the name supplied, this may be positional matching using celestial or Cartesian coordinates, exact matching on the value of a string column, or other things. A list and explanation of the available matching algorithms is given in Section 7.1. The value supplied for this parameter determines the meanings of the values required by the `params`, `values*` and `tuning` parameter(s).

[Default: `sky`]

`ocmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section

6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`
- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

`out = <out-table>` ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `-`]

`params = <match-params>` ***(String[])***

Determines the parameters of this match. This is typically one or more tolerances such as error radii. It may contain zero or more values; the values that are required depend on the match type selected by the `matcher` parameter. If it contains multiple values, they must be separated by spaces; values which contain a space can be 'quoted' or "quoted".

`progress = none|log|profile` ***(String)***

Determines whether information on progress of the match should be output to the standard error stream as it progresses. For lengthy matches this is a useful reassurance and can give guidance about how much longer it will take. It can also be useful as a performance diagnostic.

The options are:

- `none`: no progress is shown
- `log`: progress information is shown
- `profile`: progress information and limited time/memory profiling information are shown

[Default: `log`]

`runner = classic|parallel|sequential|partest` ***(RowRunner)***

Selects the threading implementation. The options are currently:

- `classic`: uses legacy sequential implementation
- `parallel`: uses multithreaded implementation for large tables
- `sequential`: uses multithreaded implementation but with only a single thread
- `partest`: uses multithreaded implementation even when tables are small

The `parallel` option should run faster than the others, at least for large matches and where multiple processing cores are available. At present however, this code is less well tested than the legacy implementation, so `classic` is the default. `sequential` and `partest` are mostly provided for testing purposes.

`parallel` ought to run correctly, but if you want to be sure of the results, you can start by comparing the results from `sequential` and `parallel` and then use `parallel` for larger matches.

Changing this parameter should make no difference to the matching results. If you notice any discrepancies **please report them**.

[Default: `classic`]

`scorecol = <col-name>` ***(String)***

Gives the name of a column in the output table to contain the "match score" for each pairwise match. The meaning of this column is dependent on the chosen `matcher`, but it typically represents a distance of some kind between the two matching points. If a null value is chosen, no score column will be inserted in the output table. The default value of this parameter depends on `matcher`.

[Default: `Score`]

`suffix1 = <label>` ***(String)***

If the `fixcols` parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from table 1.

[Default: `_1`]

`suffix2 = <label>` ***(String)***

If the `fixcols` parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from table 2.

[Default: `_2`]

`tuning = <tuning-params>` ***(String[])***

Tuning values for the matching process, if appropriate. It may contain zero or more values; the values that are permitted depend on the match type selected by the `matcher` parameter. If it contains multiple values, they must be separated by spaces; values which contain a space can be 'quoted' or "quoted". If this optional parameter is not supplied, sensible defaults will be

chosen.

values1 = <expr-list> ***(String[])***

Defines the values from table 1 which are used to determine whether a match has occurred. These will typically be coordinate values such as RA and Dec and perhaps some per-row error values as well, though exactly what values are required is determined by the kind of match as determined by `matcher`. Depending on the kind of match, the number and type of the values required will be different. Multiple values should be separated by whitespace; if whitespace occurs within a single value it must be 'quoted' or "quoted". Elements of the expression list are commonly just column names, but may be algebraic expressions calculated from zero or more columns as explained in Section 10.

values2 = <expr-list> ***(String[])***

Defines the values from table 2 which are used to determine whether a match has occurred. These will typically be coordinate values such as RA and Dec and perhaps some per-row error values as well, though exactly what values are required is determined by the kind of match as determined by `matcher`. Depending on the kind of match, the number and type of the values required will be different. Multiple values should be separated by whitespace; if whitespace occurs within a single value it must be 'quoted' or "quoted". Elements of the expression list are commonly just column names, but may be algebraic expressions calculated from zero or more columns as explained in Section 10.

B.35.2 Examples

Here are some examples of using `tmatch2`:

```
stilts tmatch2 in1=obs_v.xml in2=obs_i.xml out=obs_iv.xml \
               matcher=sky values1="ra dec" values2="ra dec" params="2"
```

Takes two input catalogues (VOTables), one with observations in the V band and the other in the I band, and performs a match to find objects within 2 arcseconds of each other. The result is a new table containing only rows where a match was found.

```
stilts tmatch2 survey.fits ifmt2=csv mycat.csv \
               icmd1='addskycoords fk4 fk5 RA1950 DEC1950 RA2000 DEC2000' \
               matcher=skyerr \
               params=10 values1="RA2000 DEC2000 POS_ERR"  values2="RA DEC 0" \
               join=2not1 omode=count
```

Here a comma-separated-values file is being compared with a FITS catalogue representing some survey results. Positions in the survey catalogue use the FK4 B1950.0 system, and so a preprocessing step is inserted to create new position columns in the first input table using the FK5 J2000.0 system, which is what the other input table uses. The survey catalogue contains a POS_ERR column which gives the positional uncertainty of its entries, so the `skyerr` matcher is used, which takes account of this; the third entry in the `values1` parameter is the POS_ERR column (in arcsec). Since the second input table has no positional uncertainty information, 0 is used as the third entry in `values2`. The `params` gives a rough idea of the scale of the object separations, but its value does not affect the result. The join type is `2not1`, which means the output table will only contain those entries which are in the second input table but not in the first one. The output table is not stored, but the number of rows it contains (the number of objects represented in the CSV file but not the survey) is written to the screen.

```
stilts tmatch2 ifmt1=ascii ifmt2=ascii in1=cat-a.txt in2=cat-b.txt \
               matcher=2d values1='X Y' values2='X Y' params=5 join=1and2 \
               suffix1=_a suffix2=_b \
               ocmd='addcol XDIFF X_a-X_b; addcol YDIFF Y_a-Y_b' \
               ocmd'keepcols "XDIFF YDIFF"' omode=stats
```

Two ASCII-format catalogues are matched, where rows are considered to match if their X,Y positions are within 5 units of each other in some Cartesian space. The result of the matching operation is a table of all the matched rows, containing columns named X_a, Y_a, X_b and Y_b (along with any others in the input tables) - the `suffix*` parameters describe how the input X and Y columns are to be renamed to avoid duplicate column names in the output table. To this result are added two new columns, representing the X and Y positional difference between the rows from one input table and those from the other. The `keepcols` filter then throws all the other columns away, retaining only these difference columns. The final two-column table is not stored anywhere, but (`omode=stats`) statistics including mean and standard deviation are calculated on its columns and displayed to the screen. Having done all this, you can examine the average X and Y differences between the two input tables for matched rows, and if they differ significantly from zero, you can conclude that there is a systematic error between the positions in the two input files.

```
stilts tmatch2 in1=mgc.fits in2=6dfgs.xml join=1and2 find=all \
               matcher=sky+1d params='3 0.5' \
               values1='ra dec bmag' values2='RA2000 DEC2000 B_MAG" \
               out=pairs.fits
```

This performs a match with a matcher that combines `sky` and `1d` match criteria. This means that the only rows which match are those which are *both* within 3 arcsec of each other on the sky *and* and within 0.5 blue magnitudes. Note that for both the `params` and the `values1` and `values2` parameters, the items for the `sky` matcher (RA and DEC) are listed first, followed by those for the `1d` matcher (in this case, blue magnitude).

B.36 `tmatchn`: Crossmatches multiple tables using flexible criteria

`tmatchn` performs efficient and flexible crossmatching between multiple tables. It can match rows on the basis of their relative position in the sky, or alternatively using many other criteria such as separation in in some isotropic or anisotropic Cartesian space, identity of a key value, or some combination of these; the full range of match criteria is dicussed in Section 7.1.

Since the match criteria define what counts as a match between two objects, it is not immediately obvious what is meant by a multi-table match. In fact the command can work in one of two distinct modes, controlled by the `multimode` parameter. In `pairs` mode, one table (by default the first input table) is designated the reference table, and pair matches between each of the other tables and that one are identified. In `group` mode groups of objects from all the input tables are identified, as discussed in Section 7.2. Currently, in both cases an output matched row cannot contain more than one object from each input table. Options for output of multiple rows per input table per match may be forthcoming in future releases if there is demand.

`tmatchn` is intended for use with more than two input tables - see `tmatch1` and `tmatch2` for 1- and 2-table crossmatching respectively.

B.36.1 Usage

The usage of `tmatchn` is

```
stilts <stilts-flags> tmatchn nin=<count> ifmtN=<in-format> inN=<tableN>
                              icmdN=<cmds> ocmd=<cmds>
                              omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|p
                              out=<out-table> ofmt=<out-format>
                              multimode=pairs|group iref=<table-index>
                              matcher=<matcher-name> params=<match-params>
                              tuning=<tuning-params> valuesN=<expr-list>
                              joinN=default|match|nomatch|always
                              fixcols=none|dups|all suffixN=<label>
                              progress=none|log|profile
```

```
runner=classic|parallel|sequential|partest
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.TableMatchN`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`fixcols = none|dups|all` ***(Fixer)***
Determines how input columns are renamed before use in the output table. The choices are:

- `none`: columns are not renamed
- `dups`: columns which would otherwise have duplicate names in the output will be renamed to indicate which table they came from
- `all`: all columns will be renamed to indicate which table they came from

If columns are renamed, the new ones are determined by `suffix*` parameters.

[Default: `dups`]

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on input table #N as specified by parameter `inN`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of input table #N as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <tableN>` ***(StarTable)***
The location of input table #N. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`iref = <table-index>` ***(Integer)***

If `multimode=pairs` this parameter gives the index of the table in the input table list which is to serve as the reference table (the one which must be matched by other tables). Ignored in other modes.

Row ordering in the output table is usually tidiest if the default setting of 1 is used (i.e. if the first input table is used as the reference table).

[Default: `1`]

`joinN = default|match|nomatch|always` ***(MultiJoinType)***

Determines which rows from input table N are included in the output table. The matching algorithm determines which of the rows in each of the input tables correspond to which rows in the other input tables, and this parameter determines what to do with that information.

The default behaviour is that a row will appear in the output table if it represents a match of rows from two or more of the input tables. This can be altered on a per-input-table basis however by choosing one of the non-default options below:

- `match`: Rows are included only if they contain an entry from input table N.
- `nomatch`: Rows are included only if they do not contain an entry from input table N.
- `always`: Rows are included if they contain an entry from input table N (overrides any match and nomatch settings of other tables).
- `default`: Input table N has no special effect on whether rows are included.

[Default: `default`]

`matcher = <matcher-name>` ***(MatchEngine)***

Defines the nature of the matching that will be performed. Depending on the name supplied, this may be positional matching using celestial or Cartesian coordinates, exact matching on the value of a string column, or other things. A list and explanation of the available matching algorithms is given in Section 7.1. The value supplied for this parameter determines the meanings of the values required by the `params`, `values*` and `tuning` parameter(s).

[Default: `sky`]

`multimode = pairs|group` ***(String)***

Defines what is meant by a multi-table match. There are two possibilities:

- `pairs`: Each output row corresponds to a single row of the *reference table* (see parameter `iref`) and contains entries from other tables which are pair matches to that. If a reference table row matches multiple rows from one of the other tables, only the best one is included.
- `group`: Each output row corresponds to a group of entries from the input tables which are mutually linked by pair matches between them. This means that although you can get from any entry to any other entry via one or more pair matches, there is no guarantee that any entry is a pair match with any other entry. No table has privileged status in this case. If there are multiple entries from a given table in the match group, an arbitrary one is chosen for inclusion (there is no unique way to select the best). See Section 7.2 for more discussion.

Note that which rows actually appear in the output is also influenced by the `joinN` parameter.

[Default: `pairs`]

`nin = <count>` ***(Integer)***

The number of input tables for this task. For each of the input tables N there will be associated parameters `ifmtN`, `inN` and `icmdN`.

`ocmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This

parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- `out`
- `meta`
- `stats`
- `count`
- `checksum`
- `cgi`
- `discard`
- `topcat`
- `samp`
- `plastic`
- `tosql`
- `gui`

Use the `help=omode` flag or see Section 6.4 for more information.

[Default: `out`]

`out = <out-table>` ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: -]

`params = <match-params>` ***(String[])***

Determines the parameters of this match. This is typically one or more tolerances such as error radii. It may contain zero or more values; the values that are required depend on the match type selected by the `matcher` parameter. If it contains multiple values, they must be separated by spaces; values which contain a space can be 'quoted' or "quoted".

`progress = none|log|profile` ***(String)***

Determines whether information on progress of the match should be output to the standard error stream as it progresses. For lengthy matches this is a useful reassurance and can give guidance about how much longer it will take. It can also be useful as a performance diagnostic.

The options are:

- `none`: no progress is shown
- `log`: progress information is shown
- `profile`: progress information and limited time/memory profiling information are shown

[Default: `log`]

`runner = classic|parallel|sequential|partest` ***(RowRunner)***
Selects the threading implementation. The options are currently:

- `classic`: uses legacy sequential implementation
- `parallel`: uses multithreaded implementation for large tables
- `sequential`: uses multithreaded implementation but with only a single thread
- `partest`: uses multithreaded implementation even when tables are small

The `parallel` option should run faster than the others, at least for large matches and where multiple processing cores are available. At present however, this code is less well tested than the legacy implementation, so `classic` is the default. `sequential` and `partest` are mostly provided for testing purposes.

`parallel` ought to run correctly, but if you want to be sure of the results, you can start by comparing the results from `sequential` and `parallel` and then use `parallel` for larger matches.

Changing this parameter should make no difference to the matching results. If you notice any discrepancies **please report them**.

[Default: `classic`]

`suffixN = <label>` ***(String)***
If the `fixcols` parameter is set so that input columns are renamed for insertion into the output table, this parameter determines how the renaming is done. It gives a suffix which is appended to all renamed columns from table N.

[Default: `_N`]

`tuning = <tuning-params>` ***(String[])***
Tuning values for the matching process, if appropriate. It may contain zero or more values; the values that are permitted depend on the match type selected by the `matcher` parameter. If it contains multiple values, they must be separated by spaces; values which contain a space can be 'quoted' or "quoted". If this optional parameter is not supplied, sensible defaults will be chosen.

`valuesN = <expr-list>` ***(String[])***
Defines the values from table N which are used to determine whether a match has occurred. These will typically be coordinate values such as RA and Dec and perhaps some per-row error values as well, though exactly what values are required is determined by the kind of match as determined by `matcher`. Depending on the kind of match, the number and type of the values required will be different. Multiple values should be separated by whitespace; if whitespace occurs within a single value it must be 'quoted' or "quoted". Elements of the expression list are commonly just column names, but may be algebraic expressions calculated from zero or more columns as explained in Section 10.

B.36.2 Examples

Here are some examples of using `tmatchn`:

```
stilts tmatchn multimode=pairs nin=4 matcher=sky params=5 \
       in1=transients.txt ifmt1=ascii values1='alpha delta' \
       in2=2mass_virgo.fits values2='ra2000 dec2000' \
       in3=sdss_virgo.fits values3='ra dec' \
       in4=first_virgo.fits values4='pos_eq_ra pos_eq_dec' \
       out=matches.xml ofmt=votable-binary
```

Compares a text-format table "transients.txt" against each of three other catalogues covering the same region of sky, and outputs a table which contains a row for each row of "transients.txt" which matches (is within 5 arcsec) of an object in any of the other tables.

```
stilts tmatchn multimode=pairs nin=4 matcher=sky params=5 \
       in1=transients.txt ifmt1=ascii suffix1='_t' values1='alpha delta' \
       in2=2mass_virgo.fits suffix2='_2mass' values2='ra2000 dec2000' \
       in3=sdss_virgo.fits  suffix3='_sdss'  values3='ra dec' \
       in4=first_virgo.fits suffix4='_first' values4='pos_eq_ra pos_eq_dec' \
       fixcols=all join1=always \
       ocmd='keepcols "*_t designation_2mass SDSSName_sdss id_field_first"' \
       out=matches.xml ofmt=votable-binary
```

Similar to the previous example but with some doctoring of what the output table will look like. The `fixcols=all` and `suffixN` assignments mean that all the columns from the input tables will be renamed for output by adding the given suffixes. The `keepcols` filter applied to the output table throws out all the columns except the ones from the reference table (`*_t`) and one column from each of the other table giving object identifiers. This output table will probably be easier to read (though contain less information) than that from the previous example). Additionally, the `join1=always` assignment means that the output table will have one row for each row of the reference table (transients.txt), even if no matches are found for it.

```
stilts tmatchn multimode=group nin=3 matcher=skyerr params=8 \
       in1=Hband.fits values1='RA DEC SEEING*2' \
       in2=Jband.fits values2='RA DEC SEEING*2' \
       in3=Kband.fits values3='RA DEC SEEING*2' \
       omode=topcat
```

Performs a group-mode match. There is no reference table, so that an output row will result for each object which is represented in any two of the input catalogues. The match takes account of per-object errors equivalent to twice the recorded seeing, which is in the region of 8 arcsec. Note that this may not operate as expected if the catalogues contain multiple distinct objects too close (in comparison to the declared separation) to each other. The resulting matched table is sent directly to TOPCAT (if available).

B.37 `tmulti`: Writes multiple tables to a single container file

`tmulti` takes multiple input tables and writes them as separate tables to a single output container file. The container file must be of some format which can contain more than one table, for instance a FITS file (which can contain multiple extensions) or a VOTable document (which can contain multiple TABLE elements). Filtering may be performed on the tables prior to writing them. It is not necessary that all the tables are similar (e.g. that they all have the same type and number of columns), but the same processing commands will be applied to all of them. For more individual control, use the `tmultin` task.

B.37.1 Usage

The usage of `tmulti` is

```
stilts <stilts-flags> tmulti in=<table> [<table> ...] ifmt=<in-format>
                            multi=true|false istream=true|false
                            icmd=<cmds> out=<out-file> ofmt=<out-format>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.MultiCopy`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`icmd = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on each input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***
Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

The same format parameter applies to all the tables specified by `in`.

[Default: `(auto)`]

`in = <table> [<table> ...]` ***(TableProducer[])***
Locations of the input tables. Either specify the parameter multiple times, or supply the input tables as a space-separated list within a single use.

The following table location forms are allowed:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

Compression in any of the supported compression formats (Unix compress, gzip or bzip2) is expanded automatically.

A list of input table locations may be given in an external file by using the indirction character '@'. Thus "`in=@filename`" causes the file `filename` to be read for a list of input table locations. The locations in the file should each be on a separate line.

`istream = true|false` ***(Boolean)***
If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource

usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

The same streaming flag applies to all the tables specified by `in`.

[Default: `false`]

`multi = true|false` ***(Boolean)***
Determines whether all tables, or just the first one, from input table files will be used. If set `false`, then just the first table from each file named by `in` will be used. If `true`, then all tables present in those input files will be used. This only has an effect for file formats which are capable of containing more than one table, which effectively means FITS and VOTable and their variants.

[Default: `false`]

`ofmt = <out-format>` ***(String)***
Specifies the format in which the output tables will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

Not all output formats are capable of writing multiple tables; if you choose one that is not, an error will result.

[Default: `(auto)`]

`out = <out-file>` ***(uk.ac.starlink.util.Destination)***
The location of the output file. This is usually a filename to write to. If it is equal to the special value "-" the output will be written to standard output.

[Default: `-`]

B.37.2 Examples

Here are some examples of using `tmulti`:

```
stilts tmulti ifmt=ascii in=t1.txt in=t2.txt in=t3.txt
              ofmt=fits out=tables.fits
```

Takes the three named ASCII format tables and writes them into a multi-extension FITS file, as three separate BINTABLE HDUs. These tables do not need to be of the same shape or otherwise similar.

```
stilts tmulti ifmt=ascii in="t1.txt t2.txt t3.txt"
              ofmt=fits out=tables.fits
```

Does exactly the same as the previous example.

```
stilts tmulti ifmt=ascii in=@inlist.lis
              ofmt=fits out=tables.fits
```

This will have the same effect as the previous two examples if a file name "inlist.lis" in the current directory contains three lines, "t1.txt", "t2.txt" and "t3.txt".

```
stilts tmulti in=extract.fits multi=true out=extract.vot
```

This takes the table extensions from a multi-extension FITS file and writes them out as a multi-TABLE VOTable document. The `multi=true` setting is required, since this means that all the tables from the input file are used as input; if it was set false, only the first table HDU

from the input file would be used.

```
stilts tmulti in=extract.fits multi=true out=extract.vot
             icmd='badval -999 *MAG'
```

Does the same as the previous example, but additionally replaces with a blank value occurrences of the value "-999" in columns whose name ends with "MAG" in any of the input tables before copying them.

B.38 `tmultin`: Writes multiple processed tables to single container file

`tmultin` takes multiple input tables and writes them to a single output container file. The container file must be of some format which can contain more than one table, for instance a FITS file (which can contain multiple extensions) or a VOTable document (which can contain multiple TABLE elements). Individual filtering may be performed on the tables prior to writing them, and their formats may be specified individually. If you want to apply the same pre-processing to all the input tables, you may find the `tmulti` command more convenient.

B.38.1 Usage

The usage of `tmultin` is

```
stilts <stilts-flags> tmultin nin=<count> ifmtN=<in-format> inN=<tableN>
                              icmdN=<cmds> out=<out-file> ofmt=<out-format>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.MultiCopyN`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`icmdN = <cmds>` ***(ProcessingStep[])***
Specifies processing to be performed on input table #N as specified by parameter `inN`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmtN = <in-format>` ***(String)***
Specifies the format of input table #N as specified by parameter `inN`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`inN = <tableN>` ***(StarTable)***
The location of input table #N. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmtN` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`nin = <count>` ***(Integer)***

The number of input tables for this task. For each of the input tables N there will be associated parameters `ifmtN`, `inN` and `icmdN`.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output tables will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

Not all output formats are capable of writing multiple tables; if you choose one that is not, an error will result.

[Default: `(auto)`]

`out = <out-file>` ***(uk.ac.starlink.util.Destination)***

The location of the output file. This is usually a filename to write to. If it is equal to the special value "-" the output will be written to standard output.

[Default: -]

B.38.2 Examples

Here are some examples of using `tmultin`:

```
stilts tmultin nin=3 in1=t1.xml ifmt1=votable
                     in2=t2.fit ifmt2=fits
                     in3=t3.txt ifmt3=ascii
                     out=tables.fits
```

Takes three input tables in different formats, and writes them out as a single multi-extension FITS file.

```
stilts tmultin nin=3 in1=data.fits icmd1='every 10;   head 100'
                     in2=data.fits icmd2='every 100;  head 100'
                     in3=data.fits icmd3='every 1000; head 100'
                     out=samples.xml ofmt=votable
```

Writes three hundred-row tables as separate TABLE elements in a single output VOTable document. Each of the output tables is a sample from the same input table, but sampled differently; the first is every tenth row, the second every hundredth, and the third every thousandth.

B.39 `tpipe`: Performs pipeline processing on a table

tpipe performs all kinds of general purpose manipulations which take one table as input. It is extremely flexible, and can do the following things amongst others:

- calculate statistics
- display metadata
- select rows in various ways, including algebraically
- define new columns as algebraic functions of old ones
- delete or rearrange columns
- sort rows
- convert between table formats

and combine these operations. You can think of it as a supercharged table copying tool.

The basic operation of tpipe is that it reads an input table, performs zero or more processing steps on it, and then does something with the output. There are therefore three classes of things you need to tell it when it runs:

Input table location
Specified by the in, ifmt and istream parameters.

Processing steps
Either provide a string giving steps as the value of one or more cmd parameters, or the name of a file containing the steps using the script parameter. The steps that you can perform are described in Section 6.1.

Output table destination
What happens to the output table is determined by the value of the omode parameter. By default, omode=out, in which case the table is written to a new table file in a format determined by ofmt. However, you can do other things with the result such as calculate the per-column statistics (omode=stats), view only the table and column metadata (omode=meta), display it directly in TOPCAT (omode=topcat) etc.

See Section 6 for a more detailed explanation of these ideas.

The parameters mentioned above are listed in detail in the next section.

B.39.1 Usage

The usage of tpipe is

```
stilts <stilts-flags> tpipe ifmt=<in-format> istream=true|false cmd=<cmds>
                            omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|pla
                            out=<out-table> ofmt=<out-format>
                            [in=]<table>
```

If you don't have the stilts script installed, write "java -jar stilts.jar" instead of "stilts" - see Section 3. The available <stilts-flags> are listed in Section 2.1. For programmatic invocation, the Task class for this command is uk.ac.starlink.ttools.task.TablePipe.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

cmd = <cmds> ***(ProcessingStep[])***
Specifies processing to be performed on the input table as specified by parameter in, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ifmt = <in-format>` ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istream = true|false` ***(Boolean)***

If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: `(auto)`]

`omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui` ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is `out`, which means that the result will be written as a new table to disk or elsewhere, as determined by the `out` and `ofmt` parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (`<mode-args>`) are required to determine the exact behaviour.

Possible values are

- out
- meta
- stats
- count
- checksum
- cgi
- discard
- topcat
- samp
- plastic
- tosql
- gui

Use the help=omode flag or see Section 6.4 for more information.

[Default: out]

out = <out-table> ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if omode has its default value of "out".

[Default: -]

B.39.2 Examples

Here are some examples of tpipe in use with explanations of what's going on. For simplicity these examples assume that you have the stilts script installed and are using a Unix-like shell; see Section 3 for an explanation of how to invoke the command if you just have the Java classes.

```
stilts tpipe cat.fits
```

Writes a FITS table to standard output in human-readable form. Since no mode specifier is given, omode=out is assumed, and output is to standard output in text format.

```
stilts tpipe cmd='head 5' cat.fits.gz
```

Does the same as the last example, but with one processing step: only the first five rows of the table are output. In this case, the input file is compressed using gzip - this is automatically detected.

```
stilts tpipe ifmt=csv xxx.csv \
             cmd='keepcols "index ra dec"' \
             omode=out ofmt=fits xxx.fits
```

Reads from a comma-separated values file, writes to a FITS file, and discards all columns in the input table apart from INDEX, RA and DEC. Note the quoting in the cmd argument: the outer quotes are so that the argument of the cmd parameter itself (keepcols "index ra dec") is not split up by spaces (to protect it from the shell), and the inner quotes are to keep the colid-list argument of the keepcols command together.

```
stilts tpipe ifmt=votable \
             cmd='addcol IV_SUM "(IMAG+VMAG)"' \
             cmd='addcol IV_DIFF "(IMAG-VMAG)"' \
             cmd='delcols "IMAG VMAG"' \
             omode=out ofmt=votable \
      < tab1.vot \
```

```
      > tab2.vot
```

Replaces two columns by their sum and difference in a VOTable. Since neither the `in` nor `out` parameters have been specified, the input and output are actually byte streams on standard input and standard output of the `tpipe` command in this case. The processing steps first add a column representing the sum, then add a column representing the difference, then delete the original columns.

```
stilts tpipe cmd='addskycoords -inunit sex fk5 gal \
                               RA2000 DEC2000 GAL_LONG GAL_LAT' \
             6dfgs.fits 6dfgs+gal.fits
```

Adds columns giving galactic coordinates to a table. Both input and output tables are FITS files. The galactic coordinates, stored in new columns named GAL_LONG and GAL_LAT, are calculated from FK5 J2000.0 coordinates given in the existing columns named RA2000 and DEC2000. The input (FK5) coordinates are represented as sexagesimal strings (hh:mm:ss, dd:mm:ss), and the output ones are numeric degrees.

```
stilts -disk tpipe 2dfgrs_ngp.fits \
                   cmd='keepcols "SEQNUM AREA ECCENT"' \
                   cmd='sort -down AREA' \
                   cmd='head 20'
```

Displays selected columns for the 20 rows with largest values in the AREA column of a FITS table. First the columns of interest are selected, then the rows are sorted into descending order by the value of the AREA column, then the first 20 rows of the resulting table are selected, and the result is written to standard output. Since a sort is being performed here, it's not possible to do all the processing a row at a time, since all the AREA values must be available for comparison during the sort. Two things are done here to accommodate this fact: first the column selection is done before the sort, so that it's only a 3-column table which needs to be available for random access, reducing the temporary storage required. Secondly the `-disk` flag is supplied, which means that temporary disk files rather than memory will be used for caching table data.

```
stilts tpipe 2dfgrs_ngp.fits \
             cmd='keepcols "SEQNUM AREA ECCENT"' \
             cmd='sorthead -down 20 AREA'
```

Has exactly the same effect as the previous example. However, the algorithm used by the `sorthead` filter is in most cases faster and cheaper on memory (only 20 rows ever have to be stored in this case), so this is generally a better approach than combining the `sort` and `head` filters.

```
stilts tpipe omode=meta cmd=@commands.lis http://archive.org/data/survey.vot.Z
```

Outputs column and table metadata about a table. In this case the table is a compressed VOTable at the end of a URL. Processing is performed according to the commands contained in a file named "commands.lis" in the current directory.

```
stilts tpipe in=survey.fits
             cmd='select "skyDistanceDegrees(hmsToDegrees(RA),dmsToDegrees(DEC), \
                                             hmsToDegrees(2,28,11),dmsToDegrees(-6,49,45)) \
                          < 5./60."' \
             omode=count
```

Counts the number of rows within a given 5 arcmin cone of sky in a FITS table. The `skyDistanceDegrees` function is an expression which calculates the distance between the position specified in a row (as given by its RA and DEC columns) and a given point on the sky (here, 02:28:11,-06:49:45). Since `skyDistanceDegrees`'s arguments and return value are in decimal degrees, some conversions are required: the RA and DEC columns are sexagesimal

strings which are converted using the `hmsToDegrees` and `dmsToDegrees` functions respectively. Different versions of these functions (ones which take numeric arguments) are used to convert the coordinates of the fixed point to degrees. The result is compared to a constant expression representing 5 arcminutes in degrees. Any rows of the input table for which this comparison is true are included in the output. An alternative function, `skyDistanceRadians` which works in radians, is also available. These functions and constants used here are described in detail in Section 10.6.5 and Section 10.6.6.

```
stilts tpipe ifmt=ascii survey.txt \
             cmd='select "OBJTYPE == 3 && Z > 0.15"' \
             cmd='keepcols "IMAG JMAG KMAG"' \
             omode=stats
```

Calculate statistics on the I, J and K magnitudes of selected objects from a catalogue. Only those rows with the given OBJTYPE and in the given Z range are included. The minimum, maximum, mean, standard deviation etc of the IMAG, JMAG and KMAG columns will be written to standard output.

```
stilts -classpath lib/drivers/mysql-connector-java.jar \
       -Djdbc.drivers=com.mysql.jdbc.Driver \
       tpipe in=x.fits cmd="explodeall" omode=tosql \
             protocol=mysql host=localhost db=ASTRO1 dbtable=TABLEX \
             write=dropcreate user=mbt
```

Writes a FITS table to an SQL table, converting array-valued columns to scalar ones. To make the SQL connection work properly, the classpath is augmented to include the path of the MySQL JDBC driver and the `jdbc.drivers` system property is set to the JDBC driver class name. The output will be written as a new table named TABLEX in the MySQL database named ASTRO1 on a MySQL server on the local host. The password, if required, will be prompted for, as would any of the other required parameters if they had not been given on the command line. Any existing table in ASTRO1 with the name TABLEX is overwritten. The only processing done here is by the `explodeall` command, which takes any columns which have fixed-size array values and replaces them in the output with multiple scalar columns.

```
java -classpath stilts.jar:lib/drivers/mysql-connector-java.jar
     -Djdbc.drivers=com.mysql.jdbc.Driver \
     uk.ac.starlink.ttools.Stilts \
     tpipe in=x.fits \
           cmd=explodeall \
           omode=out \
           out="jdbc:mysql://localhost/ASTRO1?user=mbt#TABLEX"
```

This does exactly the same as the previous example, but achieves it in a slightly different way. In the first place, java is invoked directly with the necessary flags rather than getting the `stilts` script to do it. Note that you cannot use java's `-jar` flag in this case, because doing it like that would not permit access to the additional classes that contain the JDBC driver. In the second place we use `omode=out` rather than `omode=tosql`. For this we need to supply an `out` value which encodes the information about the SQL connection and table in a special URL-like format. As you can see, this is a bit arcane, which is why the `omode=tosql` mode can be a help.

```
stilts tpipe USNOB.FITS cmd='every 1000000' omode=stats
```

Calculates statistics on a selection of the rows in a catalogue, and writes the result to the terminal. In this example, every millionth row is sampled.

B.40 `tskymap`: Calculates sky density maps

`tskymap` calculates a weighted density map (or, to put it another way, a histogram) on the sky from

columns of an input table. The sky is divided up into some discrete set of tiles according to a specified tessellation scheme (currently HEALPix or HTM are supported), and the required quantities are aggregated into bins corresponding to these tiles. The output table has a column giving the pixel index identifying each tile, plus one or more columns each representing an aggregation of a quantity from the input table.

By default the number of rows from the input table falling within each tile is included as the first column in the output table. But by specifying the `cols` and `combine` parameters you can add more columns giving the sum, mean, median or other statistics of input table columns or expressions as well.

The output table can then, for instance, be plotted using `plot2sky`'s `healpix` layer type (though an alternative is to do that plot directly using a `skydensity` layer).

In the case of HEALPix tiling, the result can also be output in a FITS file suitable for use by external applications that understand the semi-standard FITS-Healpix convention. Note in this case, for maximum compatibility, the `fits-healpix` output format should in general be used.

See also `tgridmap`, which does the same thing for N-dimensional grid geometry.

B.40.1 Usage

The usage of `tskymap` is

```
stilts <stilts-flags> tskymap ifmt=<in-format> istream=true|false
                              icmd=<cmds> ocmd=<cmds>
                              omode=out|meta|stats|count|checksum|cgi|discard|topcat|samp|p
                              out=<out-table> ofmt=<out-format>
                              lon=<expr/deg> lat=<expr/deg>
                              tiling=hpx<K>|healpixnest<K>|healpixring<K>|htm<K>
                              count=true|false
                              cols=<expr>[;<combiner>[;<name>]] ...
                              combine=sum|sum-per-unit|count|count-per-unit|mean|median|Q1|
                              perunit=steradian|degree2|arcmin2|arcsec2|mas2|uas2
                              complete=true|false
                              [in=]<table>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.SkyDensityMap`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`cols = <expr>[;<combiner>[;<name>]] ...` ***(String[])***

Defines the quantities to be calculated. The value is a space-separated list of items, one for each aggregated column in the output table.

Each item is composed of one, two or three tokens, separated by semicolon (";") characters:

- `<expr>`: *(required)* column name or expression using the expression language for the quantity to be aggregated.
- `<combiner>`: *(optional)* combination method, using the same options as for the `combine` parameter. If omitted, the value specified for that parameter will be used.
- `<name>`: *(optional)* name of output column; if omitted, the `<expr>` value (perhaps somewhat sanitised) will be used.

It is often sufficient just to supply a space-separated list of input table column names for this parameter, but the additional syntax may be required for instance if it's required to calculate both a sum and mean of the same input column.

combine =
sum|sum-per-unit|count|count-per-unit|mean|median|Q1|Q3|min|max|stdev|hit
(Combiner)

Defines the default way that values contributing to the same density map bin are combined together to produce the value assigned to that bin. Possible values are:

- `sum`: the sum of all the combined values per bin
- `sum-per-unit`: the sum of all the combined values per unit of bin size
- `count`: the number of non-blank values per bin (weight is ignored)
- `count-per-unit`: the number of non-blank values per unit of bin size (weight is ignored)
- `mean`: the mean of the combined values
- `median`: the median
- `Q1`: first quartile
- `Q3`: third quartile
- `min`: the minimum of all the combined values
- `max`: the maximum of all the combined values
- `stdev`: the sample standard deviation of the combined values
- `hit`: 1 if any values present, NaN otherwise (weight is ignored)

For density-like values (`count-per-unit`, `sum-per-unit`) the scaling is additionally influenced by the `perunit` parameter.

Note this value may be overridden on a per-column basis by the `cols` parameter.

[Default: `mean`]

complete = true|false ***(Boolean)***

Determines whether the output table contains a row for every pixel in the tiling, or only the rows for pixels in which some of the input data fell.

The value of this parameter may affect performance as well as output. If you know that most pixels on the sky will be covered, it's probably a good idea to set this true, and if you know that only a small patch of sky will be covered, it's better to set it false.

[Default: `false`]

count = true|false ***(Boolean)***

Controls whether a COUNT column is added to the output table along with any other columns that may have been requested. If included, this reports the number of rows from the input table that fell within the corresponding bin.

[Default: `true`]

icmd = <cmds> ***(ProcessingStep[])***

Specifies processing to be performed on the input table as specified by parameter `in`, before any other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

ifmt = <in-format> ***(String)***

Specifies the format of the input table as specified by parameter `in`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program

will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in = <table>` ***(StarTable)***

The location of the input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`istream = true|false` ***(Boolean)***

If set true, the input table specified by the `in` parameter will be read as a stream. It is necessary to give the `ifmt` parameter in this case. Depending on the required operations and processing mode, this may cause the read to fail (sometimes it is necessary to read the table more than once). It is not normally necessary to set this flag; in most cases the data will be streamed automatically if that is the best thing to do. However it can sometimes result in less resource usage when processing large files in certain formats (such as VOTable). This parameter is ignored for scheme-specified tables.

[Default: `false`]

`lat = <expr/deg>` ***(String)***

Latitude in degrees for the position of each row in the input table. This may simply be a column name, or it may be an algebraic expression as explained in Section 10. The sky system used here will determine the grid on which the output map is built.

`lon = <expr/deg>` ***(String)***

Longitude in degrees for the position of each row in the input table. This may simply be a column name, or it may be an algebraic expression as explained in Section 10. The sky system used here will determine the grid on which the output map is built.

`ocmd = <cmds>` ***(ProcessingStep[])***

Specifies processing to be performed on the output table, after all other processing has taken place. The value of this parameter is one or more of the filter commands described in Section 6.1. If more than one is given, they must be separated by semicolon characters (";"). This parameter can be repeated multiple times on the same command line to build up a list of processing steps. The sequence of commands given in this way defines the processing pipeline which is performed on the table.

Commands may alteratively be supplied in an external file, by using the indirection character '@'. Thus a value of "`@filename`" causes the file `filename` to be read for a list of filter commands to execute. The commands in the file may be separated by newline characters and/or semicolons, and lines which are blank or which start with a '`#`' character are ignored.

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 - matching is case-insensitive and you can use just the first few letters). If it has the special value "`(auto)`" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if `omode` has its default value of "`out`".

[Default: (auto)]

omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is out, which means that the result will be written as a new table to disk or elsewhere, as determined by the out and ofmt parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (<mode-args>) are required to determine the exact behaviour.

Possible values are

- out
- meta
- stats
- count
- checksum
- cgi
- discard
- topcat
- samp
- plastic
- tosql
- gui

Use the help=omode flag or see Section 6.4 for more information.

[Default: out]

out = <out-table> ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if omode has its default value of "out".

[Default: -]

perunit = steradian|degree2|arcmin2|arcsec2|mas2|uas2 ***(SolidAngleUnit)***

Defines the unit of sky area used for scaling density-like combinations (e.g. combine=count-per-unit or sum-per-unit). If the combination mode is calculating values per unit area this configures the area scale in question. For non-density-like combination modes (e.g. combine=sum or mean) it has no effect.

Possible values are:

- steradian: steradian
- degree2: square degree
- arcmin2: square arcminute
- arcsec2: square arcsecond
- mas2: square milliarcsec
- uas2: square microarcsec

[Default: degree2]

tiling = hpx<K>|healpixnest<K>|healpixring<K>|htm<K> ***(SkyTiling)***

Describes the sky tiling scheme that is in use. One of the following values may be used:

- hpxK: alias for healpixnestK
- healpixnestK: HEALPix using the Nest scheme at order K
- healpixringK: HEALPix using the Ring scheme at order K
- htmK: Hierarchical Triangular Mesh at level K

So for instance `hpx5` or `healpixnest5` would both indicate the HEALPix NEST tiling scheme at order 5.

At level K, there are 12*4^K HEALPix pixels, or 8*4^K HTM pixels on the sky. More information about these tiling schemes can be found at the HEALPix and HTM web sites.

[Default: `hpx5`]

B.40.2 Examples

Here are some examples of using `tskymap`:

```
stilts tskymap in=iras_psc.fits lon=RA lat=DEC out=iras_map.csv
```

Writes a table representing a density map of IRAS point sources on the sky. The output is a 2-column CSV file: the first column is pixel index, and the second column is the number of rows in the input table whose (RA,DEC) positions fall within that pixel. The default tiling is used (currently level 5 HEALPix, which has 12288 pixels).

```
stilts tskymap in=iras_psc.fits lon=RA lat=DEC tiling=hpx6
               ocmd='healpixmeta -csys C'
               ofmt=fits-healpix out=iras_map.fits
```

This does a similar job to the previous example, but explicitly requests HEALPix level 6 tiling, and outputs to a FITS variant that other applications such as Aladin understand as containing a HEALPix map. The output format is specified with `ofmt=fits-healpix`, which writes a FITS file following the HEALPix-FITS convention. An additional filter "`healpixmeta -csys C`" is also supplied here: this indicates that the HEALPix pixels in the output file are to be considered in the **C**elestial (=equatorial) sky system. If this is not done, Aladin assumes the coordinate system to be Galactic and will align the results incorrectly on the sky.

```
stilts tskymap in=2mpz.fits icmd='addskycoords fk5 galactic ra dec glon glat'
               lon=glon lat=glat tiling=hpx6
               cols='jCorr-hCorr hCorr-kCorr jCorr-kCorr' combine=median
               count=false
               complete=true
               out=2mpzColors.fits
```

Writes a table with columns giving median J-H, H-K and J-K colours, but no source count column, from the sources in the file 2mpz.fits, aggregated over each tile of a level 6 HEALPix grid. The input table has equatorial coordinates, but the `addskycoords` filter has been used so that the grid is laid on the sky with galactic coordinates. Setting `complete=true` guarantees that a row is written to the output file for every sky pixel, including empty ones. Note in this case that the output format is not specified explicitly, so it will be inferred from the filename, to be "normal" FITS, rather than `healpix-fits`. The output data will still be present, but external applications may not automatically identify the pixel column.

```
stilts tskymap in=tgas_source.fits tiling=hpx7 lon=l lat=b count=false
               cols=phot_g_n_obs combine=sum-per-unit perunit=arcmin2
               ofmt=fits-healpix ocmd='healpixmeta -csys G'
               out=obs-density.fits
```

Calculates a level-7 HEALPix map where each cell contains the local number of observations per square arcminute, as determined by summing all the `phot_g_n_obs` values from the input table that fall into each tile and applying a suitable scaling factor. The grid coordinate system is explicitly labelled as **G**alactic.

```
stilts tskymap in=gaia_source.colfits tiling=hpx8 lon=l lat=b count=true
               cols='bp_rp;mean phot_bp_mean_flux;sum phot_rp_mean_flux;sum'
```

```
                out=gaia-stats.fits
```

Simultaneously calculates the mean value of the `bp_rp` column and the summed values of the `phot_bp_mean_flux` and `phot_rp_mean_flux` columns for each HEALPix level 8 tile, as well as the source density.

```
stilts tskymap in=gums_lmc.fits lon=alpha lat=delta
               tiling=hpx14 complete=false omode=count
```

This prepares a source count map at healpix level 14 (3 billion pixels) from the given input table. Since `complete=false`, only rows for non-empty pixels are included in the output table. Then, since the output mode is `count`, these rows are just counted, discarding the pixels themselves, giving the number of level-14 healpix pixels touched by the sources in this input file. Note this is not necessarily the most efficient way to calculate coverage information.

B.41 `tskymatch2`: Crossmatches 2 tables on sky position

`tskymatch2` performs a crossmatch of two tables based on the proximity of sky positions. You specify the columns or expressions giving right ascension and declination in degrees for each input table, and a maximum permissible separation in arcseconds, and the resulting joined table is output.

If you omit expressions for the RA and Dec, an attempt is made to identify the columns to use using column Unified Content Descriptors (UCDs) or names. First columns bearing appropriate UCD1 or UCD1+ values (`POS_EQ_RA`, `POS_EQ_RA_MAIN`, `pos.eq.ra` or `pos.eq.ra;meta.main` and their equivalents for declination) are sought. If these cannot be found, columns named something like "RA" or "RA2000" are sought. If either is found, the column units are consulted and radian->degree conversions are performed if necessary (degrees are assumed if no unit value is given). If nothing likely can be found, then the command will fail with an error message. This search logic is intended as a convenience only; it is somewhat ad hoc and subject to change. To make sure that the correct angle values are being used, specify the `ra` and `dec` position parameters explicitly.

`tskymatch2` is simply a cut-down version, provided for convenience, of the more general matching task `tmatch2`. If you want more match options or otherwise more configurability, you can probably find it by using `tmatch2`.

B.41.1 Usage

The usage of `tskymatch2` is

```
stilts <stilts-flags> tskymatch2 ifmt1=<in-format> ifmt2=<in-format>
                                 omode=out|meta|stats|count|checksum|cgi|discard|topcat|sam
                                 out=<out-table> ofmt=<out-format>
                                 ra1=<expr> dec1=<expr> ra2=<expr>
                                 dec2=<expr> error=<value/arcsec>
                                 tuning=<healpix-k>
                                 join=1and2|1or2|all1|all2|1not2|2not1|1xor2
                                 find=all|best|best1|best2
                                 runner=classic|parallel|sequential|partest
                                 [in1=]<table1> [in2=]<table2>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.SkyMatch2`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`dec1 = <expr>` ***(String)***

Declination in degrees for the position of each row of table 1. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

`dec2 = <expr>` ***(String)***
Declination in degrees for the position of each row of table 2. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

`error = <value/arcsec>` ***(Double)***
The maximum separation permitted between two objects for them to count as a match. Units are arc seconds.

`find = all|best|best1|best2` ***(PairMode)***
Determines what happens when a row in one table can be matched by more than one row in the other table. The options are:

- `all`: All matches. Every match between the two tables is included in the result. Rows from both of the input tables may appear multiple times in the result.
- `best`: Best match, symmetric. The best pairs are selected in a way which treats the two tables symmetrically. Any input row which appears in one result pair is disqualified from appearing in any other result pair, so each row from both input tables will appear in at most one row in the result.
- `best1`: Best match for each Table 1 row. For each row in table 1, only the best match from table 2 will appear in the result. Each row from table 1 will appear a maximum of once in the result, but rows from table 2 may appear multiple times.
- `best2`: Best match for each Table 2 row. For each row in table 2, only the best match from table 1 will appear in the result. Each row from table 2 will appear a maximum of once in the result, but rows from table 1 may appear multiple times.

The differences between `best`, `best1` and `best2` are a bit subtle. In cases where it's obvious which object in each table is the best match for which object in the other, choosing betwen these options will not affect the result. However, in crowded fields (where the distance between objects within one or both tables is typically similar to or smaller than the specified match radius) it will make a difference. In this case one of the asymmetric options (`best1` or `best2`) is usually more appropriate than `best`, but you'll have to think about which of them suits your requirements. The performance (time and memory usage) of the match may also differ between these options, especially if one table is much bigger than the other.

[Default: `best`]

`ifmt1 = <in-format>` ***(String)***
Specifies the format of the first input table as specified by parameter `in1`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`ifmt2 = <in-format>` ***(String)***
Specifies the format of the second input table as specified by parameter `in2`. The known formats are listed in Section 5.1.1. This flag can be used if you know what format your table is in. If it has the special value `(auto)` (the default), then an attempt will be made to detect the format of the table automatically. This cannot always be done correctly however, in which case the program will exit with an error explaining which formats were attempted. This parameter is ignored for scheme-specified tables.

[Default: `(auto)`]

`in1 = <table1>` ***(StarTable)***

The location of the first input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt1` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`in2 = <table2>` ***(StarTable)***

The location of the second input table. This may take one of the following forms:

- A filename.
- A URL.
- The special value "`-`", meaning standard input. In this case the input format must be given explicitly using the `ifmt2` parameter. Note that not all formats can be streamed in this way.
- A scheme specification of the form `:<scheme-name>:<scheme-args>`.
- A system command line with either a "`<`" character at the start, or a "`|`" character at the end ("`<syscmd`" or "`syscmd|`"). This executes the given pipeline and reads from its standard output. This will probably only work on unix-like systems.

In any case, compressed data in one of the supported compression formats (gzip, Unix compress or bzip2) will be decompressed transparently.

`join = 1and2|1or2|all1|all2|1not2|2not1|1xor2` ***(JoinType)***

Determines which rows are included in the output table. The matching algorithm determines which of the rows from the first table correspond to which rows from the second. This parameter determines what to do with that information. Perhaps the most obvious thing is to write out a table containing only rows which correspond to a row in both of the two input tables. However, you may also want to see the unmatched rows from one or both input tables, or rows present in one table but unmatched in the other, or other possibilities. The options are:

- `1and2`: An output row for each row represented in both input tables (INNER JOIN)
- `1or2`: An output row for each row represented in either or both of the input tables (FULL OUTER JOIN)
- `all1`: An output row for each matched or unmatched row in table 1 (LEFT OUTER JOIN)
- `all2`: An output row for each matched or unmatched row in table 2 (RIGHT OUTER JOIN)
- `1not2`: An output row only for rows which appear in the first table but are not matched in the second table
- `2not1`: An output row only for rows which appear in the second table but are not matched in the first table
- `1xor2`: An output row only for rows represented in one of the input tables but not the other one

[Default: `1and2`]

`ofmt = <out-format>` ***(String)***

Specifies the format in which the output table will be written (one of the ones in Section 5.1.2 -

matching is case-insensitive and you can use just the first few letters). If it has the special value "(auto)" (the default), then the output filename will be examined to try to guess what sort of file is required usually by looking at the extension. If it's not obvious from the filename what output format is intended, an error will result.

This parameter must only be given if omode has its default value of "out".

[Default: (auto)]

omode = out|meta|stats|count|checksum|cgi|discard|topcat|samp|plastic|tosql|gui ***(ProcessingMode)***

The mode in which the result table will be output. The default mode is out, which means that the result will be written as a new table to disk or elsewhere, as determined by the out and ofmt parameters. However, there are other possibilities, which correspond to uses to which a table can be put other than outputting it, such as displaying metadata, calculating statistics, or populating a table in an SQL database. For some values of this parameter, additional parameters (<mode-args>) are required to determine the exact behaviour.

Possible values are

- out
- meta
- stats
- count
- checksum
- cgi
- discard
- topcat
- samp
- plastic
- tosql
- gui

Use the help=omode flag or see Section 6.4 for more information.

[Default: out]

out = <out-table> ***(TableConsumer)***

The location of the output table. This is usually a filename to write to. If it is equal to the special value "-" (the default) the output table will be written to standard output.

This parameter must only be given if omode has its default value of "out".

[Default: -]

ra1 = <expr> ***(String)***

Right ascension in degrees for the position of each row of table 1. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

ra2 = <expr> ***(String)***

Right ascension in degrees for the position of each row of table 2. This may simply be a column name, or it may be an algebraic expression calculated from columns as explained in Section 10. If left blank, an attempt is made to guess from UCDs, column names and unit annotations what expression to use.

runner = classic|parallel|sequential|partest ***(RowRunner)***

Selects the threading implementation. The options are currently:

- classic: uses legacy sequential implementation
- parallel: uses multithreaded implementation for large tables
- sequential: uses multithreaded implementation but with only a single thread

- `partest`: uses multithreaded implementation even when tables are small

The `parallel` option should run faster than the others, at least for large matches and where multiple processing cores are available. At present however, this code is less well tested than the legacy implementation, so `classic` is the default. `sequential` and `partest` are mostly provided for testing purposes.

`parallel` ought to run correctly, but if you want to be sure of the results, you can start by comparing the results from `sequential` and `parallel` and then use `parallel` for larger matches.

Changing this parameter should make no difference to the matching results. If you notice any discrepancies **please report them**.

[Default: `classic`]

`tuning = <healpix-k>` ***(Integer)***

Tuning parameter that controls the pixel size used when binning the rows. The legal range is from 0 (corresponding to pixel size of about 60 degrees) to 20 (about 0.2 arcsec). The value of this parameter will not affect the result but may affect the performance in terms of CPU and memory resources required. A default value will be chosen based on the size of the `error` parameter, but it may be possible to improve performance by adjusting the default value. The value used can be seen by examining the progress output. If your match is taking a long time or is failing from lack of memory it may be worth trying different values for this parameter.

B.41.2 Examples

Here are some examples of using `tskymatch2`:

```
stilts tskymatch2 in1=obs_v.xml in2=obs_i.xml out=obs_iv.xml \
                  ra1=OBS_RA dec1=OBS_DEC ra2=OBS_RA dec2=OBS_DEC error=2
```

Takes two input catalogues (VOTables), one with observations in the V band and the other in the I band, and performs a match to find objects within 2 arcseconds of each other. The result is a new VOTable containing only rows where a match was found.

```
stilts tskymatch2 in1=obs_v.xml in2=obs_i.xml out=obs_iv.xml \
                  error=2
```

This is the same as the previous example but without explicit specification of the sky position columns in either table. It will work only if those columns are identified with appropriate UCDs, for instance `pos.eq.ra;meta.main` and `pos.eq.dec:meta.main`. If no suitable UCDs are in place this invocation will fail with an error.

```
stilts tskymatch2 in1=virgo1.txt ifmt1=ascii in2=mgc.fits \
                  ra1='radiansToDegrees(raRad)' dec1='radiansToDegrees(deRad)' \
                  ra2=MGC_ALPHA_J2000 dec2=MGC_DELTA_J2000 \
                  error=10 join=2not1 omode=count
```

Object positions in the text file virgo1.txt are compared to those in the FITS file mgc.fits. The angles have been recorded in the text file in radians, so they are converted to degrees here before use. Use of the `join=2not1` parameter causes the command to identify all the objects in the first list which do not have counterparts within 10 arcsec in the second list. The number of such objects found is simply output to the terminal.

B.42 `votcopy`: Transforms between VOTable encodings

The VOTable standard provides for three basic encodings of the actual data within each table:

TABLEDATA, BINARY and FITS. TABLEDATA is a pure-XML encoding, which is relatively easy for humans to read and write. However, it is verbose and not very efficient for transmission and processing, for which reason the more compact BINARY format has been defined. FITS format shares the advantages of BINARY, but is more likely to be used where a VOTable is providing metadata 'decoration' for an existing FITS table. In addition, the BINARY and FITS encodings may carry their data either inline (as the base64-encoded text content of a `STREAM` element) or externally (referenced by a `STREAM` element's `href` attribute).

These different formats have their different advantages and disadvantages. Since, to some extent, programmers are humans too, much existing VOTable software deals in TABLEDATA format even though it may not be the most efficient way to proceed. Conversely, you might wish to examine the contents of a BINARY-encoded table without use of any software more specialised than a text editor. So there are times when it is desirable to convert from one of these encodings to another.

`votcopy` is a tool which translates between these encodings while making a minimum of other changes to the VOTable document. The processing may result in some changes to lexical details such as whitespace in start tags, but the element structure is not modified. Unlike `tpipe` it does not impose STIL's model of what constitutes a table on the data between reading it in and writing it out, so subtleties dependent on the exact structure of the VOTable document will not be mangled. The only important changes should be the contents of `DATA` elements in the document.

B.42.1 Usage

The usage of `votcopy` is

```
stilts <stilts-flags> votcopy version=1.0|1.1|1.2|1.3|1.4
                              charset=<xml-encoding> cache=true|false
                              href=true|false nomagic=true|false
                              base=<location>
                              [in=]<location> [out=]<location>
                              [format=]TABLEDATA|BINARY|BINARY2|FITS
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.VotCopy`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`base = <location>` ***(String)***
Determines the name of external output files written when the `href` flag is true. Normally these are given names based on the name of the output file. But if this flag is given, the names will be based on the `<location>` string. This flag is compulsory if `href` is true and `out=-` (output is to standard out), since in this case there is no default base name to use.

`cache = true|false` ***(Boolean)***
Determines whether the input tables are read into a cache prior to being written out. The default is selected automatically depending on the input table; so you should normally leave this flag alone.

`charset = <xml-encoding>` ***(Charset)***
Selects the Unicode encoding used for the output XML. The available options are dependent on your JVM, use `help=charset` for a full listing. Setting the value null will use the JVM's system default.

[Default: `UTF-8`]

`format = TABLEDATA|BINARY|BINARY2|FITS` ***(uk.ac.starlink.votable.DataFormat)***
Determines the encoding format of the table data in the output document. If `null` is selected,

then the tables will be data-less (will contain no DATA element), leaving only the document structure. Data-less tables are legal VOTable elements.

The `BINARY2` format is only available for `version=1.3`

[Default: `TABLEDATA`]

`href = true|false` ***(Boolean)***
In the case of BINARY or FITS encoding, this determines whether the STREAM elements output will contain their data inline or externally. If set false, the output document will be self-contained, with STREAM data inline as base64-encoded characters. If true, then for each TABLE in the document the binary data will be written to a separate file and referenced by an href attribute on the corresponding STREAM element. The name of these files is usually determined by the name of the main output file; but see also the `base` flag.

`in = <location>` ***(String)***
Location of the input VOTable. May be a URL, filename, or "-" to indicate standard input. The input table may be compressed using one of the known compression formats (Unix compress, gzip or bzip2).

[Default: -]

`nomagic = true|false` ***(Boolean)***
Eliminate the `null` attributes of `VALUES` elements where they are no longer required. In VOTable versions <=1.2, the only way to specify null values for integer-type scalar columns was to use the `null` attribute of the `VALUES` element to indicate an in-band magic value representing null. From VOTable v1.3, null values can be represented using empty `<TD>` elements or flagged specially in `BINARY2` streams. In these cases, it is recommended (though not required) not to use the `VALUES`/`null` mechanism.

If this parameter is set true, then any `VALUES`/`null` attributes will be removed in VOTable 1.3 BINARY2 or TABLEDATA output. If this results in an empty `VALUES` element, it too will be removed.

This parameter is ignored if the output VOTable version is lower than 1.3 or if `format`=BINARY/FITS.

[Default: `true`]

`out = <location>` ***(String)***
Location of the output VOTable. May be a filename or "-" to indicate standard output.

[Default: -]

`version = 1.0|1.1|1.2|1.3|1.4` ***(uk.ac.starlink.votable.VOTableVersion)***
Determines the version of the VOTable standard to which the output will conform. If null (the default), the output table will have the same version as the input table.

B.42.2 Examples

Normal use of `votcopy` is pretty straightforward. We give here a couple of examples of its input and output.

Here is an example VOTable document, `cat.vot`:

```
<VOTABLE>
<RESOURCE>

<TABLE name="Authors">
<FIELD name="AuthorName" datatype="char" arraysize="*"/>
<DATA>
<TABLEDATA>
<TR><TD>Charles Messier</TD></TR>
```

```
<TR><TD>Mark Taylor</TD></TR>
</TABLEDATA>
</DATA>
</TABLE>

<RESOURCE>
<COOSYS equinox="J2000.0" epoch="J2000.0" system="eq_FK4"/>
<TABLE name="Messier Objects">
<FIELD name="Identifier" datatype="char" arraysize="10"/>
<FIELD name="RA" datatype="double" units="degrees"/>
<FIELD name="Dec" datatype="double" units="degrees"/>
<DATA>
<TABLEDATA>
<TR> <TD>M51</TD> <TD>202.43</TD> <TD>47.22</TD> </TR>
<TR> <TD>M97</TD> <TD>168.63</TD> <TD>55.03</TD> </TR>
</TABLEDATA>
</DATA>
</TABLE>
</RESOURCE>

</RESOURCE>
</VOTABLE>
```

Note that it contains more structure than just a flat table: there are two `TABLE` elements, the `RESOURCE` element of the second one being nested in the `RESOURCE` of the first. Processing this document using a generic table tool such as `tpipe` or `tcopy` would lose this structure.

To convert the data encoding to BINARY format, we simply execute

```
stilts votcopy format=binary cat.vot
```

and the output is

```
<?xml version="1.0"?>
<VOTABLE>
<RESOURCE>

<TABLE name="Authors">
<FIELD name="AuthorName" datatype="char" arraysize="*"/>
<DATA>
<BINARY>
<STREAM encoding='base64'>
AAAAD0NoYXJsZXMgTWVzc2llcgAAAAtNYXJrIFRheWxvcg==
</STREAM>
</BINARY>
</DATA>
</TABLE>

<RESOURCE>
<COOSYS equinox="J2000.0" epoch="J2000.0" system="eq_FK4"/>
<TABLE name="Messier Objects">
<FIELD name="Identifier" datatype="char" arraysize="10"/>
<FIELD name="RA" datatype="double" units="degrees"/>
<FIELD name="Dec" datatype="double" units="degrees"/>
<DATA>
<BINARY>
<STREAM encoding='base64'>
TTUxAAAAAAAAAEBpTcKPXCj2QEecKPXCj1xNOTcAAAAAAAAAQGUUKPXCj1xAS4PX
Cj1wpA==
</STREAM>
</BINARY>
</DATA>
</TABLE>
</RESOURCE>

</RESOURCE>
</VOTABLE>
```

Note that both tables in the document have been translated to BINARY format. The basic structure of the document is unchanged: the only differences are within the `DATA` elements. If we ran

```
stilts votcopy format=tabledata
```

on either this output or the original input then the output would be identical (apart perhaps from whitespace) to the input table, since the data are originally in TABLEDATA format.

To generate a VOTable document with the data in external files, the `href` parameter is used. We will output in FITS format this time. Executing:

```
stilts votcopy format=fits href=true cat.vot fcat.vot
```

writes the following to the file `fcat.vot`:

```
...
<DATA>
<FITS>
<STREAM href="fcat-1.fits"/>
</FITS>
</DATA>
...
<DATA>
<FITS>
<STREAM href="fcat-2.fits"/>
</FITS>
</DATA>
...
```

(the unchanged parts of the document have been skipped here for brevity). The actual data are written in two additional files in the same directory as the output file, `fcat-1.fits` and `fcat-2.fits`. These filenames are based on the main output filename, but can be altered using the `base` flag if required. Note this has also given you FITS binary table versions of all the tables in the input VOTable document, which can be operated on by normal FITS-aware software quite separately from the VOTable if required.

B.43 `votlint`: Validates VOTable documents

The VOTable standard, while not hugely complicated, has a number of subtleties and it's not difficult to produce VOTable documents which violate it in various ways. In some cases the errors are small and a parser is likely to process the document without noticing the trouble. In other cases, the errors are so serious that it's hard for any software to make sense of it. In many cases in between, different software will react in different ways, in the worst case appearing to parse a VOTable but in fact understanding the wrong data.

`votlint` is a program which can check a VOTable document and spot places where it does not conform to the VOTable standard, or places which look like they may not mean what the author intended. It is meant for use in two main scenarios:

1. For authors of VOTables and VOTable-producing software, to check that the documents they produce are legal and problem-free.
2. For users of VOTables (including authors of VOTable-processing software) who are having problems with one and want to know whether it is the data or the software at fault.

Validating a VOTable document against the VOTable schema or DTD of course goes a long way towards checking a VOTable document for errors, but it by no means does the whole job, simply because the schema/DTD specification languages don't have the facilities to understand the data structure of a VOTable document. For instance the VOTable schema will allow any plain text content in a `TD` element, but whether this makes sense in a VOTable depends on the `datatype` attribute of the corresponding `FIELD` element. There are many other examples. `votlint` tackles this by parsing the VOTable document in a way which understands its structure and assessing the content as critically as it can. For any incorrect or questionable content it finds, it will output a short message describing the problem and giving its location in the document. What you do with this information is then up to you.

Using votlint is very straightforward. The votable argument gives the location (filename or URL) of a VOTable document. Otherwise, the document will be read from standard input. Error and warning messages will be written on standard output. Each message is prefixed with the location at which the error was found (if possible the line and column are shown, though this is dependent on your JVM's default XML parser). If multiple instances of the same problem are found, by default only a few repeats of the message are reported; this can be controlled with the maxrepeat parameter. The processing is SAX-based, so arbitrarily long tables can be processed without heavy memory use.

votlint can't guarantee to pick up every possible error in a VOTable document, but it ought to pick up many of the most serious errors that are commonly made in authoring VOTables.

Note: votlint's handling of XML namespaces seems to be somewhat dependent on the XML parser in use. As far as I can see, Crimson (the default in many JREs) works for any namespace arrangements, but Xerces seems to have problems when validating documents which use namespace prefixes. Not sure about other parsers. This probably won't cause you trouble, but if it does you may need to set validate=false to work around it. Contact the author if this seems to be a serious issue for you.

B.43.1 Usage

The usage of votlint is

```
stilts <stilts-flags> votlint ucd=true|false unit=true|false
                              maxrepeat=<int-value> validate=true|false
                              version=1.0|1.1|1.2|1.3|1.4 out=<location>
                              [votable=]<location>
```

If you don't have the stilts script installed, write "java -jar stilts.jar" instead of "stilts" - see Section 3. The available <stilts-flags> are listed in Section 2.1. For programmatic invocation, the Task class for this command is uk.ac.starlink.ttools.task.VotLint.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

maxrepeat = <int-value> ***(Integer)***
Puts a limit on the number of times that the same issue will be reported. With this set to a relatively small number, the output is not cluttered with many repetitions of the same problem.

[Default: 4]

out = <location> ***(uk.ac.starlink.util.Destination)***
Destination file for output messages. May be a filename or "-" to indicate standard output.

[Default: -]

ucd = true|false ***(Boolean)***
If true, the ucd attributes on FIELD and PARAM elements etc are checked for conformance against the UCD1+ standard or a list of known UCD1 terms.

[Default: true]

unit = true|false ***(Boolean)***
If true, the unit attributes on FIELD and PARAM elements are checked for conformance against the VOUnits standard; if false, no such checks are made.

The VOTable 1.4 standard recommends use of VOUnits, though there are some inconsistencies in the text on this topic. Earlier VOTable versions refer to a different (CDS) unit syntax, which is not checked by votlint. So by default unit syntax is checked when the VOTable is 1.4 or greater, and not for earlier versions, but that can be overridden by giving a

`true` or `false` value for this parameter.

The wording of the VOTable and VOUnit standards do not strictly require use of VOUnit syntax even at VOTable 1.4, so failed checks result in Warning rather than Error reports.

`validate = true|false` ***(Boolean)***
Whether to validate the input document aganist the VOTable DTD. If true (the default), then as well as `votlint`'s own checks, it is validated against an appropriate version of the VOTable DTD which picks up such things as the presence of unknown elements and attributes, elements in the wrong place, and so on. Sometimes however, particularly when XML namespaces are involved, the validator can get confused and may produce a lot of spurious errors. Setting this flag false prevents this validation step so that only `votlint`'s own checks are performed. In this case many violations of the VOTable standard concerning document structure will go unnoticed.

[Default: `true`]

`version = 1.0|1.1|1.2|1.3|1.4` ***(uk.ac.starlink.votable.VOTableVersion)***
Selects the version of the VOTable standard which the input table is supposed to exemplify. The version may also be specified within the document using the "version" attribute of the document's VOTABLE element; if it is and it conflicts with the value specified by this flag, a warning is issued.

If no value is provided for this parameter (the default), the version will be determined from the VOTable itself.

`votable = <location>` ***(InputStream)***
Location of the VOTable to be checked. This may be a filename, URL or "-" (the default), to indicate standard input. The input may be compressed using one of the known compression formats (Unix compress, gzip or bzip2).

B.43.2 Items Checked

Votlint checks that the XML input is well-formed, and, unless the `valid=false` parameter is supplied, that it validates against the 1.0 DTD or 1.1, 1.2, 1.3 or 1.4 schema as appropriate. Some of the validity checks are also done by `votlint` internally, so that some validity-type errors may give rise to more than one warning. In general, the program errs on the side of verbosity.

In addition to these checks, the following checks are carried out, and lead to ERROR reports if violations are found:

- `TD` contents incompatible `datatype`/`arraysize` attributes declared in `FIELD`
- BINARY/BINARY2 data streams which don't match metadata declared in `FIELD`
- `PARAM` values incompatible with declared `datatype`/`arraysize`
- Meaningless `arraysize` declarations
- Array-valued `TD` elements with the wrong number of elements
- Array-valued `PARAM` values with the wrong number of elements
- `nrows` attribute on `TABLE` element different from the number of rows actually in the table
- `VOTABLE` `version` attribute is unknown
- `ref` attributes without matching `ID` elements elsewhere in the document
- Same `ID` attribute value on multiple elements.
- Attributes defined by external vocabularies have permitted values; hard-coded lists of known permitted values are stored internally, but the external vocabularies are consulted if necessary to check for values added after build time (currently only applies to `TIMESYS` attributes).
- Values marked by `xtype` values as defined in DALI are checked for formal compliance with DALI requirements.

Additionally, the following conditions, which are not actually forbidden by the VOTable standard, will generate WARNING reports. Some of these may result from harmless constructions, but it is wise at least to take a look at the input which caused them:

- Wrong number of `TD` elements in row of `TABLEDATA` table
- Mismatch between VOTable and FITS column metadata for FITS data encoding
- `TABLE` with no `FIELD` elements
- Use of deprecated attributes
- `FIELD` or `PARAM` elements with `datatype` of either `char` or `unicodeChar` and undeclared `arraysize` - this is a common error which can result in ignoring all but the first character in `TD` elements from a column
- `ref` attributes which reference other elements by `ID` where the reference makes no, or questionable sense (e.g. `FIELDref` references `FIELD` in a different table)
- Multiple sibling elements (such as `FIELD`s, though not `INFO`s) with the same `name` attributes
- `TIMESYS` elements never referenced.
- `ucd` attribute values are not legal UCD1+ strings (if parameter `ucd=true`)
- `unit` attribute values are not legal VOUnit strings (if parameter `unit=true`)

B.43.3 Examples

Here is a brief example of running `votlint` against a (very short) imperfect VOTable document. If the document looks like this:

```
<VOTABLE version="1.1">
 <RESOURCE>
  <TABLE nrows="2">
   <FIELD name="Identifier" datatype="char"/>
   <FIELD name="RA" datatype="double"/>
   <FIELD name="Dec" datatype="double"/>
   <DESCRIPTION>A very small table</DESCRIPTION>
   <DATA>
    <TABLEDATA>
     <TR>
      <TD>Fomalhaut</TD>
      <TD>344.48</TD>
      <TD>-29.618</TD>
      <TD>HD 216956</TD>
     </TR>
    </TABLEDATA>
   </DATA>
  </TABLE>
 </RESOURCE>
</VOTABLE>
```

then the output of a `votlint` run looks like this:

```
INFO (l.4): No arraysize for character, FIELD implies single character
ERROR (l.7): Element "TABLE" does not allow "DESCRIPTION" here.
WARNING (l.11): Characters after first in char scalar ignored (missing arraysize?)
WARNING (l.15): Wrong number of TDs in row (expecting 3 found 4)
ERROR (l.18): Row count (1) not equal to nrows attribute (2)
```

(Note that the details of the reports will vary according to the XML parser/validator that forms part of your Java installation.)

Note also that the warning at line 11 has resulted from the same error as the one at line 4 - because the `FIELD` element has no `arraysize` attribute, `arraysize="1"` (single character) is assumed, while the author almost certainly intended `arraysize="*"` (unknown length string).

By examining these warnings you can see what needs to be done to fix this table up. Here is what it should look like:

```
<VOTABLE version="1.1">
```

```
<RESOURCE>
 <TABLE nrows="1">                                    <!-- change row count -->
  <DESCRIPTION>A very small table</DESCRIPTION>       <!-- move DESCRIPTION -->
  <FIELD name="Identifier" datatype="char"
                           arraysize="*"/>            <!-- add arraysize -->
  <FIELD name="RA" datatype="double"/>
  <FIELD name="Dec" datatype="double"/>
  <DATA>
   <TABLEDATA>
    <TR>
     <TD>Fomalhaut</TD>
     <TD>344.48</TD>
     <TD>-29.618</TD>
    </TR>                                             <!-- remove extra TD -->
   </TABLEDATA>
  </DATA>
 </TABLE>
</RESOURCE>
</VOTABLE>
```

When fed this version, `votlint` gives no warnings.

B.44 `xsdvalidate`: Validates against XML Schema

`xsdvalidate` is a utility for validating XML documents against XML Schemas. This is not very specific to STILTS, and does not claim to do a better job than other XSD validators out there, so if you can find another one that suits your purposes, please use it. It was added to the package however since it seems hard to find suitable tools that do this job.

The schema is not specified by runtime parameters, elements in a given XML document are validated against any schemas associated with the namespaces defined in that document. However, it is possible to override the locations of schemas using runtime parameters. By using the `schemaloc` parameter, you can tell the validator to validate against a local copy of a schema or one at a given URL, rather than the copy that is referenced by `xsi:schemaLocation` attributes in the document itself. That can be convenient when developing a schema with a public namespace that is fixed, but content which is still subject to change. Moreover, some local copies of W3C and IVOA schemas are kept within the application, so that external network accesses are not required during validation.

By default, successful validation results in no output and a success exit status, but if there are validation errors then some indication of what went wrong is printed to standard output and the command exits with an error. This makes it suitable for use in makefiles.

B.44.1 Usage

The usage of `xsdvalidate` is

```
stilts <stilts-flags> xsdvalidate schemaloc=<namespace>=<location> ...
                                  topel=[{<ns-uri>}][<local-name>]
                                  verbose=true|false uselocals=true|false
                                  [doc=]<location>
```

If you don't have the `stilts` script installed, write "`java -jar stilts.jar`" instead of "`stilts`" - see Section 3. The available `<stilts-flags>` are listed in Section 2.1. For programmatic invocation, the Task class for this command is `uk.ac.starlink.ttools.task.XsdValidate`.

Parameter values are assigned on the command line as explained in Section 2.3. They are as follows:

`doc = <location>` ***(InputStream)***
Location of XML document to validate.

schemaloc = <namespace>=<location> ... ***(String[])***

Assignments of override schema locations to XML namespaces. One or more assignments may be supplied, each of the form `<namespace>=<location>` where the location may be a filename or URL. Multiple assignments may be made by supplying the parameter multiple times, or using a space character as a separator.

Each assignment causes any reference to the given namespace in the validated document to be validated with reference to the XSD schema at the given location rather than to a schema acquired in the default way (using `xsi:schemaLocation` attributes or using the namespace as a retrieval URL).

topel = [{<ns-uri>}][<local-name>] ***(String)***

Local name of the top-level element expected in the parsed document. If the actual parsed top-level element has a local name differing from this, an error will be reported. If no value is specified (the default) no checking is done.

uselocals = true|false ***(Boolean)***

Whether to use local copies of VO schemas where available. If true, copies of some IVOA schemas stored within the application are used instead of retrieving them from their http://www.ivoa.net/ URLs. Setting this true is generally faster and more robust against network issues, though it may risk retrieving out of date copies of the schemas.

[Default: `false`]

verbose = true|false ***(Boolean)***

If true, some INFO reports will be displayed alongside any ERROR reports resulting from the parse. This may be useful for diagnosis or reassurance.

[Default: `false`]

B.44.2 Examples

Here are some examples of using `xsdvalidate`:

```
stilts xsdvalidate example-voresource.xml
```

Validates the given file against whatever schemas it declares. Success is indicated with a non-error exit.

```
stilts xsdvalidate
       doc=example-voresource.xml
       schemaloc='http://www.ivoa.net/xml/VOResource/v1.0=vor-local.xsd'
       topel='{http://www.ivoa.net/xml/RegistryInterface/v1.0}Resource'
       verbose=true
```

Validates the named file against whatever schemas it declares, but uses the local file "`vor-local.xsd`" in the current directory as the schema associated with the "`http://www.ivoa.net/xml/VOResource/v1.0`" namespace, rather than using any schema location given in the validated file. It also requires that the top-level element has the given URI and local name; if not, the validation will fail. The `verbose=true` parameter provides some diagnostics to standard output, including indications of which schemas were used and where they were acquired from.

C Release Notes

This is STILTS, Starlink Tables Infrastructure Library Tool Set. It is a collection of non-GUI utilites for general purpose table manipulation.

Author
Mark Taylor (Bristol University)

Email
m.b.taylor@bristol.ac.uk

WWW
http://www.starlink.ac.uk/stilts/

User comments, suggestions, requests and bug reports to the above address are welcomed.

C.1 Acknowledgements

The initial development of STILTS was done under the UK's Starlink project (1980-2005, R.I.P.). Since then it has been supported by grant PP/D002486/1 from the UK's Particle Physics and Astronomy Research Council, the VOTech project (from EU FP6), the AstroGrid project (from PPARC/STFC), the AIDA project (from EU FP7), grants ST/H008470/1, ST/I00176X/1, ST/J001414/1, ST/L002388/1, ST/M000907/1, ST/R000700/1 and ST/S001980/1 from the UK's Science and Technology Facilities Council (STFC), the GAVO project (BMBF Bewilligungsnummer 05A08VHA), the European Space Agency, and the EU FP7-2013-Space project GENIUS. All of this support is gratefully acknowledged.

Apart from the excellent Java 2 Standard Edition itself, the following external libraries provide important parts of STILTS's functionality:

- JEL (GNU) for algebraic expression evaluation
- cds-healpix-java (CDS) for HEALPix manipulation
- iText (1T3XT BVBA) for PDF output
- JFreeSVG for SVG output
- EPSGraphics2D (Jibble) for encapsulated postscript output
- MOC (CDS) for Multi-Order Coverage map manipulation
- ADQL (CDS) for ADQL parsing in TAP query preparation
- Skyview in a Jar (NASA) for sky axis drawing
- JLaTeXMath (Scilab) for LaTeX typesetting in plots
- GifEncoder (Acme) for GIF output
- HTM (Sloan Digital Sky Survey) for HTM-based celestial sphere row matching (now deprecated within STILTS)

Thanks in particular to Nickolai Kouropatkine and Chris Stoughton of Fermilab for writing the PixTools specially for use in STIL.

Many people have contributed ideas and advice to the development of STILTS and its related products. A list of some of them can be found in the TOPCAT user document, SUN/253.

If you use this software in published work, the following citation would be appreciated:

> 2006ASPC..351..666T: M. B. Taylor, "STILTS - A Package for Command-Line Processing of Tabular Data", in Astronomical Data Analysis Software and Systems XV, eds. C. Gabriel et al., ASP Conf. Ser. 351, p. 666 (2006)

C.2 Version History

Releases to date have been as follows:

Version 0.1b (29 April 2005)
First public release

Version 0.2b (30 June 2005)

- Added Times func class for MJD-ISO8601 time conversions.
- Fixed bug when doing NULL_ test expressions on first column in table.

Version 1.0b (30 September 2005)
This is the first non-experimental release of STILTS, and it incorporates major changes and backward incompatibilities since version 0.2b.

Parameter system
The parameter system has undergone a complete rewrite; there is now only a single command "`stilts`", invoked using the `stilts` script or the `stilts.jar` jar file, and the various tasks are named as subsequent arguments on the command line. Command arguments are supplied after that. The new invocation syntax is described in detail elsewhere in this document. As well as invocation features such as improved on-line help, optional prompting, parameter defaulting, and more uniform access to common features, this will make it more straightforward to wrap these tasks for use in non-command-line environments, such as behind a SOAP or CORBA interface, or in a CEA-like execution environment.

Crossmatching
A new command `tmatch2` has been introduced. This provides flexible and efficient crossmatching between two input tables. Future releases will provide commands for intra-table and multi-table matching.

Concatentation
A new command `tcat` has been introduced, which allows two tables to be glued together top-to-bottom. This is currently working but very rudimentary - improvements will be forthcoming in future releases.

Calculator
A new utility command `calc` has been introduced, which performs one-line expression evaluations from the command line.

Pipeline filters
The following new filter commands for use in `tpipe` and other commands have been introduced:

- `addskycoords`: calculates new celestial coordinate pair from existing ones (FK4, FK5, ecliptic, galactic, supergalactic)
- `replacecol`: replaces column data, using existing metadata
- `badval`: replaces given 'magic' value with null
- `replaceval`: replaces given 'magic' value with any specified value
- `tablename`: edits table name
- `explodecols` and `explodecols` commands replace `explode`

The new `stream` parameter of `tpipe` now allows you to write filter commands in an external file, to facilitate more manageable command lines.

Wildarding for column specification is now allowed for some filter commands.

Algebraic functions

- New functions for converting time values between different coordinate systems (Modified Julian Date, ISO-8601, Julian Epoch and Besselian Epoch).
- New RANDOM special function.

Documentation

SUN/256 has undergone many changes. Much of the tool documentation is now automatically generated from the code itself, which goes a long way to ensuring that the documentation is correct with respect to the current state of the code.

Version 1.0-1b (7 October 2005)

Fixed jar file manifest bug which prevented working on Java 1.5

Version 1.1 (10 May 2006)

A number of new features and capabilities have been introduced:

`tcube` Command

The new tcube (Appendix B.30) command calculates N-dimensional histograms (density maps) from N columns of an input table and writes the result to a FITS file.

Processing Filters

The following new filters have been added:

- `stats` filter provides the same information as the old `stats` output mode, but allows much more flexible use of the results. It can also calculate many new quantities, including quantiles, skew and kurtosis.
- `meta` filter provides the same information as the old `meta` output mode, but allows much more flexible use of the results.
- `assert` filter provides in-pipeline logical assertions.
- `uniq` filter collapses multiple adjacent identical or similar rows.
- `sorthead` filter provides a (usually) more efficient method of doing what you could previously do by combining `sort` and `head` filters.
- `colmeta` filter adds/modifies metadata for selected columns.
- `check` filter checks table in stream - for debugging purposes only.

Additionally usage of the `sort` filter has been changed so that it can now do everything that `sortexpr` used to be able to do; `sortexpr` is now withdrawn.

Output Modes

The following new output modes have been introduced:

- `plastic` mode broadcasts the table to one or all registered PLASTIC listeners.
- `cgi` mode writes the table to standard output in a form suitable for output from a CGI script.
- `discard` mode throws away the table.

and usage of the following has been modified:

- `topcat` mode now attempts to use PLASTIC (amongst other methods) to contact TOPCAT.
- `stats` and `meta` modes are mildly deprecated in favour of the corresponding new filters (see above).

Other new features

- New IPAC table format input handler added.
- New `csv-noheader` format variant output handler added.
- `roundDecimal` and `formatDecimal` functions introduced for more control over visual appearance of numeric values.
- Experimental facilities for automatically generating a CEA application description file.

Bug fixes and minor improvements

- Now copes with 'K'-format FITS binary table columns (64-bit integers).

- Improved, though still imperfect, retention of table-wide metadata in VOTables.
- Distinctions between null and false values in boolean columns are handled more carefully for FITS and VOTable files.
- Efficiency improvement when writing FITS-plus format (now only requires a maximum of two passes rather than three of the input rows).
- Added the `mark.workaround` system property which can optionally work around a bug in some input streams ("Resetting to invalid mark" errors).
- Fixed a bug in Cartesian matching which failed to match if the required error in any dimension was zero.
- Fixed erroneous reports about unknown `ucd` and `utype` attributes of TABLE element in `votlint`.
- When joining tables, column name comparison to determine whether deduplication is required is now case-insensitive.
- Error message improved when no automatic format detection is attempted for streamed tables.
- Setting `istream=true` is now less likely to cause a "Can't re-read stream" error.

Version 1.2 (7 July 2006)

Column-oriented Storage

New features for permitting column-oriented storage (`colfits` format, new `startable.storage` policy "`sideways`") have been introduced. These can provide considerable efficiency improvements for certain tasks when working with very large (and especially wide) tables.

New VO commands

Added two new commands for querying Virtual Observatory services:

- `multicone` - Makes multiple cone search queries to the same service
- `regquery` - Queries the VO registry

These tasks are experimental and may be modified or renamed in future releases.

Other items

- `transpose` filter added.
- Added flux conversion functions (Jansky<->magnitude).
- ISO-8601 strings now permit times of 24:00:00 as they should.

Version 1.2-1 (3 August 2006)

- Tab-Separated Table (TST) format now supported for reading and writing.
- New `setparam` and `clearparams` filters.
- Added ICRS coordinate system for `addskycoords`.
- TUCDnn header cards now used in FITS files to transmit UCDs (non-standard mechanism).
- Efficiency improvements for column-oriented access.

Version 1.3 (5 October 2006)

Table Concatenation

The old `tcat` command has been replaced by more capable `tcat` and `tcatn` commands. Between them these provide concatenation of an unlimited number of homogeneous or heterogeneous input tables. Additional columns may be added to indicate which of the input tables given output rows originated from.

Parameter value indirection

Certain parameters (`in` in `tcat`, `cmd` and friends) may now be specified in the form "@filename". This indicates that the value for the parameter is to be obtained by reading

it from the named file. This is useful if a very long value is required for the parameter in question. The `script` parameter of `tpipe` has therefore been withdrawn, since it did just the same thing.

MySpace access

Direct access to the MySpace virtual file system is now provided by use of `ivo:-` or `myspace:`-type URLs.

Conversion functions

- Time conversion functions between MJD and Decimal Year have been added (Section 10.6.20).
- `toHex` and `fromHex` numeric conversion functions have been added (Section 10.6.4).

Documentation improvements

- The HTML version of SUN/256 now uses CSS to provide better highlighting of `commands` etc.
- The Output Modes and Processing Filter sections are now split into subsections to make the table of contents clearer.
- The Command Reference section now has only one level of subsection listed in the table of contents to make it clearer.

Other new features and improvements

- Added `-J` flag to `stilts` script for passing flags directly to Java.
- Added new `out` parameter to `votlint`.
- Added `-ifndim` and `-ifshape` flags to explodeall filter.
- The `exact` match mode in `tmatch2` now copes with array-valued columns.
- Added `force` parameter to `multicone` task as a workaround for some broken services.
- Added Sample (as opposed to Population) Standard Deviation/Variance calculation options to the `stats` filter.
- Improved CEA description file output - now contains details of all tasks rather than just a few, as well as various improvements in documentation etc.

Bug fixes

- Fixed erroneous complaints from `votlint` about `utype` attribute on RESOURCE elements.
- Fixed a couple of minor crossmatching bugs (which wouldn't have affected results).

Version 1.3-1 (Starlink Hokulei release)

- New command `tjoin` introduced.
- Output to MySpace can now be streamed, if running under J2SE1.5 or later.
- Slight changes to parameters for `votlint` and `votcopy`.
- Fixed bug in handling of single quotes in FITS file metadata.
- Added `-bench` flag to `stilts` command.
- Various scalability improvements for use with very large (Tb?) files.
- Improved efficiency for `text` and `ascii` output formats (now one-pass not two-pass).
- Improved CEA app-description file, including especially option lists for things like input and output formats.
- Added README.cea file to distribution.
- Fixed problem which could mis-report VOTable out of memory errors as Broken Pipe.
- Added Vega<->AB magnitude conversion constants to Fluxes functions.
- Added `duptag` parameters to tmatch2 task for customised renaming of columns with duplicated names.

- Added hyperbolic trig functions to Maths class (`sinh`, `cosh`, `tanh` and inverses).
- Added cosmology distance calculations in class Distances.
- Added `funcs` task, a browser for expression language function documentation.
- Added `-checkversion` to list of `stilts` flags.

Version 1.3-2 (6 July 2007)

- Added optional `table` parameter to `calc` command (for access to table parameters).
- Can use table parameter names in expressions using `param$` notation (Section 10.2).
- Can reference columns/parameters by UCD by using `ucd$` notation in expressions (Section 10.1) and as column identifiers (Section 6.2).
- Improved deduplication of column names when joining tables.
- Fix error in output of FITS table `TNULL` *n* header cards - write them as numeric not string values.
- Improve error message for broken CSV files.
- Modified JDBC handling so that MySQL and PostgreSQL do not run out of heap memory when streaming large datasets for input. Think I've done the same for SQL Server, but this is not tested.
- Improve error reporting in the presence of a deficient JVM (such as GNU `gcj`).
- Add locale-specific `formatDecimalLocal` functions in class Formats.
- Add `fluxToLuminosity` and `luminosityToFlux` functions in class Fluxes.
- Fix bug which was causing NullPointerExceptions in the `transpose` filter.

Version 1.3-3 (4 Sep 2007)

- Experimental, and currently undocumented, `sqlcone` task introduced, along with some classes in package `uk.ac.starlink.ttools.cone` designed for library use by AstroGrid DSA code.
- CEA description of `tmatch2 params` parameter now has `minoccurs=0`, since that can be true for exact matches.

Version 1.3-4 (10 Sep 2007)

- Fixed VotCopy bug.

Version 1.3-5 (30 Oct 2007)

- Added `-stdout` and `-stderr` flags to `stilts` command.
- Some bugs fixed in generation of CEA `app-description.xml` file.
- Documentation provided for `sqlcone` command.
- Fixed error in `fluxToLuminosity` function.

Version 1.4 (6 December 2007)

Table joins

This version provides more cross matching functionality. Added to the existing `tmatch2` command are new tasks:

- `tskymatch2`: stripped down version of `tmatch2` for ease of use when matching with sky coordinates.
- `tmatch1`: internal matcher, finds groups of objects within a table.
- `tmatchn`: finds group or multiple-pair matches between multiple (>2) tables.

Two tasks have been renamed for improved clarity and consistency:

- `multicone` is now named `coneskymatch`
- `sqlcone` is now named `sqlskymatch`

There has also been some enhancement and rationalisation of parameters for all table join tools (`tmatch*` as well as `tjoin`, `coneskymatch` and `sqlskymatch`):

- All table join commands now use similar `fixcols` and `suffix*` parameters to control renaming of duplicated columns in output tables (note this replaces the old `duptag*` parameters in `tmatch2`).
- Crossmatching tasks have a new `progress` parameter which allows you to configure whether progress is reported to the console.
- The `copycols` parameter of `coneskymatch` and `sqlskymatch` now defaults to `"*"` (include all columns from input table in the output).

Section 7 of the manual has been somewhat rearranged and improved.

Other enhancements

- FITS reader now imports table HDU header cards as table parameters.
- CeaWriter can now output CEA service definition XML config file as well as app-description file (experimental - may be withdrawn).

Bug fixes

- Embedded spaces in output ASCII format table column names are now substituted with underscores.
- Fix a bug which caused an infinite number of dots to be printed when attempting a crossmatch with an empty input table.
- Corrected `votlint` handling of TABLEDATA-type multi-dimensional `char`/`unicodeChar` arrays. These are now split up into strings by counting characters rather than using whitespace delimiters. I *think* it's doing the right thing now.

Version 1.4-1 (28 January 2008)

New RDBMS-related features

- New command `sqlclient`, which is a general JDBC-based SQL command-line client.
- New command `sqlupdate`, which allows updates to existing rows in SQL tables.
- Some changes to `tosql` output mode:
  - choice of options for how to write to the database output table, controlled by new associated parameter `write` (can be `create`, `dropcreate` or `append`)
  - associated parameter `newtable` renamed `dbtable`
  - associated parameter `database` renamed `db` for consistency with other commands

Local and service-based matching command enhancements

- New parameter `scorecol` added to `tmatch2`, `coneskymatch` and `sqlskymatch` commands, which controls adding a new column to match output tables containing a goodness-of-match value.
- New parameter `parallel` added to `coneskymatch` task which allows multiple cone searches to be carried out in parallel.
- New parameter `erract` added to `coneskymatch` which controls response to isolated failures in individual cone search queries.

General improvements

- Improved error reporting (reasons for errors are now reported even without the `-debug` flag).
- Add new help option `help='*'` which prints help for all parameters of a task at once.
- Added (mostly undocumented) `+verbose` flag for reducing verbosity level.
- Minor improvements to CEA app-description.
- Downgraded from WARNING to INFO log messages about the (extremely

common) VOTable syntax error of omitting a FIELD/PARAM element's `datatype` attribute.

Version 1.4-2 (26 March 2008)

Minor enhancements:

- Add `progress` parameter to `tmatchn`.
- Add `emptyok` parameter to `coneskymatch`.

Bugfixes:

- Fixed pair matching performance bug (slower if tables were not given in the right order) introduced at v1.4.
- Fixed null handling error in `calc` task.
- Fixed error in `stats` filter cardinality value calculation.
- Fixed minor bugs in suffix addition for matching commands `fixcols`.
- Removed unformatted XML output in `stats` filter usage message.
- Try to avoid exponential format in cone search URLs (some endpoints seem to require fixed point format).
- Minor CEA fixes.

Version 2.0b (23 October 2008)

This version contains two new major items, plotting and server mode. Both work, but are missing desirable features and have not had extensive testing in the field, so should be considered experimental at this stage.

Plotting

Two table plotting commands are now provided:

- `plot2d`: Old-style 2D Scatter Plot
- `plot3d`: Old-style 3D Scatter Plot
- `plothist`: Old-style Histogram

See also the new Plotting (Section 9) section in the manual.

Server/Servlet Mode

A new command `server` is provided which allows STILTS commands to be executed via HTTP. One purpose of this is to facilitate server-side use of the plotting commands co-located with data to generate on-the-fly graphical summaries of server-held datasets.

Smaller enhancements and bugfixes

- Efficiency improvements (~25%? in both CPU time and memory usage) for HEALPix-based sky crossmatching (thanks to Nikolay Kouropatkine at Fermilab for a new version of the PixTools library).
- New class Arrays added to algebraic functions.
- New Appendix Commands by Category (Appendix A) added to manual.
- Add `minReal` and `maxReal` functions (max/min ignoring blank values) in class Arithmetic.
- Sexagesimal field identification for ASCII input files is less stringent (now permits minutes or seconds equal to 60).
- Minor CEA fixes.
- HEALPix bug fix (PixTools bug fix update).
- Fix bug in use of `tcat`'s `loccol` parameter.

Version 2.0-1 (23 December 2008)

- Can reference columns/parameters by Utype by using `utype$` notation in expressions (Section 10.1) and as column identifiers (Section 6.2).
- Non-alphanumeric column names may now be used for algebraic expressions in the special case that the expression is just the value of a single column.
- `regquery` command has changed in implementation, data access, and output format. It now queries VOResource1.0 registries rather than the very out of date registry protocol which was used in earlier versions.

Version 2.0-2 (9 January 2009)

- Added new samp output mode which passes the generated table to clients using the SAMP protocol.
- Updated the topcat output mode to use SAMP as one way of communicating with a running TOPCAT.
- `-version` flag now reports starjava subversion revision as well as other items.

Version 2.0-3 (27 March 2009)

- Fits BINTABLE TZERO/TSCAL value reading improvements:
  - Columns with integer TZERO values now read as integers rather than floating point values where possible. This includes unsigned longs ('K'), which were previously represented as doubles with lost precision. Unsigned longs which are too large however ($>2^{63}$) are read as nulls.
  - Byte-valued columns can now be written out by `fits-basic` output handler as signed byte values (TFORM=B,TZERO=-128) rather than signed shorts (TFORM=I).
  - More comprehensive testing.
  - Fixed bug in calculating value scaled double ('D') values.
  - Fixed bug in typing value for scaled float ('E') arrays.
  - Fixed bug which caused registry queries (`regquery`) to fail for Java 1.6.
- Fix minor bugs in detail of `votlint`'s validation tests (VOTABLE element content model, INFO and PARAM and FIELD required attributes).
- Report application name and version in User-Agent header of outgoing HTTP requests.
- The fixed length Substring Array Convention for string arrays (`TFORMnn=rAw`) is now understood for FITS binary tables.
- Minor SAMP bugs fixed (JSAMP upgraded to 0.3-1).

Version 2.0-4 (17 July 2009)

- Work around J2SE mark/reset bug when loading table direct from URL.
- Produce null rather than nonsense results from sky coordinate conversions with unphysical latitudes (`addskycoords` filter).
- Produce null rather than questionable results from sexagesimal conversions with mins/secs out of range.
- Fix two bugs in `votcopy`: XML processing instructions garbled on output, and pathnames in `base` parameters inappropriately flattened in hrefs attribute values.

Version 2.0-5 (2 Oct 2009)

- VOTable 1.2 supported.
- `votlint` can now validate VOTable documents following the (provisional, 2009-09-29 PR) VOTable 1.2 standard.
- Namespacing of VOTable documents made more intelligent, and configurable using the `votable.namespacing` system property.
- `votlint` now checks that the correct XML namespaces are in use.
- Be more careful in XML, including VOTable, output; fix VOTable output encoding to be UTF-8, and ensure no illegal XML characters are written.

- HTML table output is now HTML 4.01 by default (includes THEAD and TBODY tags).
- `parse*` string->numeric conversion functions now cope with leading or trailing whitespace.
- Work around illegally truncated type declarations in IPAC tables.
- Fix a bug which caused the first table in a multi-table file (FITS or VOTable) to be used in streaming mode, even if a subsequent one was requested.
- Bug fixed in crossmatching output: entries which should have been null were sometimes written as non-null (typically large negative numbers) in FITS and in non-TABLEDATA VOTable output. This affected cells in otherwise non-nullable columns where the entire row was absent. The previous behaviour is not likely to have been mistaken for genuine results.

Version 2.1 (6 November 2009)

- `coneskymatch` can now match using SIA and SSA services as alternatives to Cone Search ones (see its new `servicetype` parameter).
- Fixed an obscure bug which could under rare circumstances cause truncation of strings with leading/trailing whitespace read from text-format files.
- A new `startable.storage` policy "`adaptive`" is now the default. This should mean running out of memory less often. The old behaviour can be restored by giving the new `-memory` command line flag.

Note that the STIL API used by this release has changed in some backwardly incompatible ways, and may change further. If you're using STILTS as a library rather than an application you might want to wait for a later release when the API has settled down.

Version 2.1-1 (21 December 2009)

- Plotting commands can now output to PDF as well as existing graphics formats.
- New filter `fixcolnames`.
- Fixed internationalisation bug which could cause `coneskymatch` to fail in locales that use "," for a decimal point.
- Significant performance improvements related to the case of VOTable documents containing many tables.

Version 2.1-2 (24 March 2010)

- JyStilts introduced. This is a jython (i.e. Python, though not CPython) interface to the STILTS commands. It is believed to be fully working, but somewhat experimental - feedback is encouraged.
- Considerable performance and scalability improvements to the crossmatching commands (`tmatch1`, `tmatch2`, `tmatchn` and `tskymatch2`). For several common regimes, using default settings, memory use has been decreased by a factor of about 5, and CPU time reduced by a factor of about 3.
- Add optional tuning parameters to crossmatch commands (parameter `tuning` for `tmatch1`, `tmatch2` and `tmatchn`, and parameter `healpixk` for `tskymatch2`). Experimentation with these can lead to significant performance improvements for given matches.
- Fixed a crossmatch bug; it was giving a possibility of suboptimal "`find=best`" match assignments when pair matching in crowded fields. Crossmatch results thus may differ between earlier versions and this one. Both are reasonable, but the newer behaviour is more correct. In non-crowded fields, there should be no change.
- Further performance improvement for VOTable documents with very many TABLEs.
- Memory management adjusted further - default (Adaptive) storage policy now uses direct allocation (`=malloc()`) for intermediate-sized buffers to avoid running out of java heap space.
- New option "`find=each`" for `coneskymatch` and `sqlskymatch` commands. This allows you to get an output table with exactly one row for each row of the input table.
- New flag `-memgui` to monitor memory usage during runs.

- Add new filter `rowrange`.
- Add new functions to Arrays: `array` functions for constructing arrays, and new aggregating functions `median` and `quantile`.
- Syntax of the crossmatching commands' `progress` parameter has changed; it now has an additional option which will write limited profiling information as well as logging as the match progresses.
- Add `ylabel` parameter to `plothist` command.
- The `random` and `sequential` filters have been renamed `randomview` and `seqview` respectively. This provides a better idea of what they do. Since they are only useful for debugging, it is unlikely that this will break anyone's existing code.
- New filter `random` introduced which converts tables to random-access if necessary.
- Document previously undocumented `legend` parameter to plotting commands.
- Matching commands `matcher` parameters can now accept classnames of `MatchEngine` implementation classes as an option.
- Classes are now distributed as a zip of jars (`stilts_jars.zip`) as an alternative to the monolithic jar file (`stilts.jar`). This may be more appropriate for those using STILTS classes in a framework that contains other third party class libraries.
- Adjusted the way that data types are read from JDBC databases. Date, Time and Timestamp typed columns will now be converted to Strings which means they can be written to most output formats (previously they were omitted from output tables).
- STILTS no longer attempts to communicate with TOPCAT using SOAP. TOPCAT's SOAP interface has been deprecated since v2.1 (2006), so this isn't likely to cause trouble, and it permits removal of SOAP (Axis) classes from the application jar file, saving several megabytes and reducing potential version clash problems.
- Fix bug in code for handling very large mapped FITS files. This was causing fatal read errors in some cases.

Version 2.2 (6 August 2010)

New capabilities for multi-table I/O have been introduced:

- New multi-table output tasks `tmulti` and `tmultin`. These currently just copy multiple input tables to a single multi-table container file (e.g. Multi-Extension FITS or multi-TABLE VOTable). Future releases may generalise the output of multi-table processing.
- New `multi` parameter introduced for `tcat` and `tmulti` tasks to pick up all tables in a multi-table container file.
- New JyStilts functions `treads` and `twrites` for multi-table I/O.

There are some additional enhancements:

- Added experimental name-resolution filter `addresolve`; this currently uses Sesame.
- Added filter `repeat`, which repeats table rows a given number of times.

And a number of bug fixes:

- Recognise unofficial column type "`long`" in IPAC format tables.
- Better behaviour (warn + failover) when attempting to read large files on 32-bit OS or JVM.
- Efficiency warning now issued for large compressed FITS files.
- Upgraded PixTools HEALPix library to 2010/02/09 version. This fixes a bug that could theoretically cause deficient crossmatch results, though I haven't managed to produce such errors.
- Fixed bug in TST table output.
- Fixed bug in FITS-plus metadata output (table parameters were getting lost).
- Corrected literature references in Fluxes conversion class documentation (thanks to Mattia Vaccari).
- Fixed bug in CSV file parsing that could ignore header row in absence of non-numeric columns.

- Shape and ElSize metadata items now correctly reported by `meta` filter.
- Fixed JyStilts bug when supplying an empty string for a parameter value.

Finally, from this release STILTS requires version 1.5 (a.k.a. 5.0) of the Java J2SE Runtime Environment; it will no longer run on version 1.4, which is now very old. I don't expect this to cause compatibility issues for anyone, but I'm interested to hear if that's not the case.

Version 2.2-1 (23 December 2010)

- Storage management improvements; removed restriction on large (>2Gb) non-FITS datasets in some circumstances.
- Efficiency improvement in sequential mapped access to large FITS files.
- Fix so FITS tables >2Gb can provide random access in 32-bit mode (though slower than 64-bit).
- FITS files now store table names in EXTNAME (and possibly EXTVAR) header cards.
- Window placement for the few GUI tasks should now behave a bit more like platform norms, rather than sitting in the top left hand corner.
- HTML table output now writes cell contents which look like URLs in HTML <A> tags.
- Basic authorization (`http://user:pass@host/path`) on table URLs handled.
- Fixed file pointer int overflow bug in FITS MultiMappedFiles.

Version 2.3 (9 May 2011)

TAP

The new commands `tapquery` and `tapresume` have been introduced. These provide support for the Table Access Protocol (TAP), and allow freeform queries in an SQL-like language to be made to remote databases.

Minor enhancements

- Random Groups HDUs are now tolerated, though not interpreted, within FITS files.
- Added `soapout` parameter to `regquery` command.
- Added `count`, `variance` and `stdev` functions to Arrays.
- Upgrade to JSAMP v1.2.
- Improve text rendering in `funcs` window display.
- Attempt case-sensitive matching before case-insensitive for column names.
- Fix `replaceval` filter to work with Infinities.

Bug fixes and workarounds

- JDBC table input handler now effectively downcasts BigInteger/BigDecimal types to Long/Double. The PostgreSQL JDBC driver seems to use the Big* types routinely for numeric values (which I don't think it used to do).
- Add workaround for J2SE bug #4795134, which could cause errors when reading compressed FITS files.
- Fix FITS character handling bug which could cause corrupted FITS files on output in presence of non-ASCII characters.
- Fix (some) JDBC connection leaks.
- Add missing parameters `dashNS` and `linewidthNS` to `plot2d` task.

Version 2.3-1 (30 June 2011)

- Added new command `taplint`. This is a validator for TAP (Table Access Protocol) services. It is only likely to be useful to people developing or operating TAP services.
- ASCII table parsers now understand python-friendly `nan` and `inf` representations.
- Added new constants to expression language `Infinity` and `NaN`.
- Fixed a significant bug in sky crossmatching. If all points in a table were on one side of the RA=0 line, but the error radius extended across that line, matches on the other side

could be missed. Matches could also be missed if different tables used different conventional ranges for RA (e.g. -180..180 in one case and 0..360 in another). This fix may in some, but not most, cases result in slower matching than previously.

- Fixed `coneskymatch` cone search verbosity parameter so that VERB=3 is not erroneously ignored.

Version 2.4 (27 October 2011)

Crossmatching:

- Two new asymmetric match options `best1` and `best2` have been added for the `find` parameter in the pair matching commands `tmatch2` and `tskymatch2`. They correspond to finding the best match in table B for each row in table A, and in crowded fields often provide more intuitive semantics than the previous symmetric `best` option (in non-crowded fields there is generally no difference). This replicates the matching performed by some other tools, including Aladin.
- New matchers have been added to permit matching of general elliptical, rather than just circular, regions in both planar and sky coordinates; see `2d_ellipse`, and `skyellipse`.
- Another new matcher is available for dealing with per-object errors in Cartesian coordinates (previously per-object errors could only be handled in sky coords); see *N* `d_err`.
- Semantics of the `skyerr` matcher have changed slightly.

Expression language functions:

- Algebraic functions involving angles are now mostly available using degrees as well as radians. The `Coords` class has been replaced by `CoordsDegrees` and `CoordsRadians` classes providing sky coordinate functions, and a new class `TrigDegrees` provides normal degree-based trigonometric functions alongside the radian-based versions in `Maths`. Some of the old function names have changed to make clear that they use radians and not degrees. This change should be much more convenient in most cases; sorry it's taken so long to get round to.
- Add new `join` function is added to the Arrays class to combine all the elements of an array into a string.

taplint:

There are several bugfixes and changes related to the TAP validator tool `taplint`, mostly thanks to bug reports etc from the TAP community:

- Improve test logic for record limiting queries.
- Errors no longer reported (e.g. E-Qxx-CNAM) for unexpected TAP_SCHEMA table column ordering (when running query stage but no metadata acquisition stages).
- Add new stage MDQ, which checks query result columns for all tables against declared metadata.
- Add check of versioned and unversioned LANG variants.
- Now uses corrected upload ID (`ivo://ivoa.net/std/TAPRegExt#upload-*`) as per most recent TAPRegExt draft.

Bug fixes and minor enhancements:

- Add parameter `parse` to `tapquery` command, allowing pre-send syntax checking of submitted ADQL.
- Add experimental system properties `star.basicauth.user` and `star.basicauth.password`.
- Improve resilience of `coneskymatch` in the presence of unreliable or inconsistent DAL services.
- A PARAMref element with no referent in a VOTable no longer causes an uncaught

NullPointerException.

Version 2.5 (28 March 2013)

New coverage-related functionality:

- Add new command `pixsample` which can sample pixel data from HEALPix table files (useful for things like Schlegel dust extinction). Also `addpixsample` filter, which does the same job.
- Add new command `pixfoot` which can generate MOC (Multi-Order Coverage) maps.
- Add MOC-based coverage filter to `coneskymatch` when using some Cone Search services (mostly VizieR). This uses the Multi-Order Coverage map service operated by CDS. It can make VizieR multi-cone queries much faster by not doing cone searches that are outside the coverage region of the catalogue in question.
- Add new class Coverage to the expression language containing MOC-related functions (currently, just `inMoc`).

Other new capabilities:

- Add IPAC table output format.
- Add new class KCorrections to the expression language, containing a method for calculating K-corrections following the method of Chilingarian and Zolotukhin.
- You can now reference tables in multi-extension FITS files by name (EXTNAME or EXTNAME-EXTVER) as an alternative to by HDU index.

VOTable enhancements:

- VOTable input, output and validation are now supported for version 1.3 of the VOTable standard.
- The version of the VOTable format used for VOTable output can now be selected, by using the system property `votable.version`. Output version is VOTable 1.2 by default.
- `votlint` has been changed so that it handles different VOTable versions more capably. Versions 1.1+ are now validated against a schema (which is how those versions are defined) rather than against a DTD hacked to do the same job as the schema. VOTable 1.3 validation is now provided.
- The `votcopy` command has a new `version` parameter to control output version, and a new `nomagic` parameter to control whether `VALUES`/`null` attributes are removed where appropriate.
- Infinite floating point values are now correctly encoded in VOTable output ("`+Inf`"/"`-Inf`", not "`Infinity`"/"`-Infinity`" as in previous versions).
- `votlint` is now stricter about floating point `TD` element contents.
- VOTable output no longer writes the `schemaLocation` attribute by default.

Other enhancements:

- Add new function `hypot` (=sqrt(x*x+y*y)) to the `Maths` class in expression language.
- Add new `split` functions for string splitting to the `Strings` class in expression language.
- Add `-utype` flags for `addcol`, `replacecol`, `colmeta` and `setparam` filters, and `utype` option for `meta` filter.
- Some changes to the `toString` function: it now works on non-numeric values, gives the right answer for `Long` integers and character values, and returns a blank value rather than the string "null" or "NaN" for blank inputs.
- Sexagesimal to numeric angle conversion functions now permit the seconds part of the sexagesimal string to be missing.

- Changes to the IPAC format definition are accommodated: the "long"/"l" type, which is apparently now official, no longer generates a warning, and headers may now use minus signs instead of whitespace.
- Add `OBS` stage (ObsTAP validation) to `taplint`.
- Add more checks to `CAP` stage of `taplint`. Declared languages (including features) and output formats are now checked.
- Tidy up error reporting a bit (fewer duplicate nested messages reported).
- PNG graphics output no longer has transparent background.
- Issue a warning for high values of `coneskymatch` `parallel` parameter.
- Upgrade JSAMP library to version 1.3-3.
- Upgrade Grégory Mantelet's ADQL library to version 1.1.

Bug fixes:

- Fix serious and long-standing bug (bad TZERO header, causes subsequent reads to fail) for FITS output of boolean array columns.
- Fix small but genuine sky matching bug. The effect was that near the poles matches near the specified threshold could be missed. The bug was in the PixTools library, fixed at the 2012-07-28 release.
- Fix bug in `tmatchn` group mode which could result in output rows with columns from only a single table, i.e. not representing an inter-table match, even when `join*=default`.
- Fix bug which failed when attempting to read FITS files with complex array columns (`TFORMn=rC/rM`).
- Fix failure when caching very large sequential tables.
- Fix bug in `replacecol` and `replaceval` filters which could cause truncation of strings in FITS and possibly VOTable output when the new value was longer than the previously declared maximum length.
- Fix `tcat`, `tcatn` so that in most cases output column metadata is compatible with all input tables, not just the first one in terms of nullability, array shape etc.
- Adjust SQL writer to avoid a type error for MySQL.
- Fix bug in HMS sexagesimal formatting: minus sign was omitted from negative angles. Now the output is forced positive.
- Cope with 1-column CSV files.
- Use the correct form "`rows`"/"`bytes`" rather than "`row`"/"`byte`" for TAP capability unit values.
- Fix error bar rendering bug which could result in diagonal lines being offset near the edge of plots.

Version 2.5-1 (1 July 2013)

New functionality

- Add read-only support for CDF (NASA Common Data Format) files.
- Add Median Absolute Deviation calculation (`MedAbsDev` and `ScMedAbsDev`) options to `stats` filter.
- Improved handling of HTTP basic authorization. 401s now generate a useful message about the `star.basicauth.*` system properties if they have not been set up.

Bug fixes and minor enhancements

- Fix CSV regression bug introduced at v2.5 - CSV files now work again with MSDOS-style line breaks.
- Fixed FITS output bug which could result in badly-formed string-valued header cards (no closing quote).
- Source code is now managed by git and not subversion. The format of the "Starjava revision" string reported by the `-version` flag has changed accordingly.

- Output mode `meta` now copes better with array-valued table parameters.
- Implemented fixes to reduce the chance of users inadvertently overloading external Cone/SIA/SSA services with multicone-like queries. First, fix it so that abandoned queries are properly terminated, rather than continuing to hit the server until completion or JVM shutdown. Second, implement a sensible default maximum value for the `parallel` parameter of `skyconematch` (though this may be adjusted with a system property).
- Quoting behaviour has changed when generating SQL to write to RDBMS tables. This ought to reduce problems related to mixed-case identifiers. However, it is possible that it could lead to unforseen new anomalies.
- More `toString` overloads - now works for byte and boolean values too.

Version 2.5-2 (7 March 2014)

- Add some more colour maps.
- Fix some broken and misdocumented non-table-output JyStilts commands (`tcube`, `pixfoot`).
- Fix bug which prevented access to long integer array elements from expression language.
- The Exact matcher now considers scalar numeric values equal if they have the same numeric value; they are no longer required to have the same type.
- Fixed a registry client bug which means that the `regquery` command can now successfully talk to the NVO/VAO/STSci registry. That has been broken since mid-2010.
- Add new command `tloop` for generating single-column tables from a numeric loop variable.
- `taplint` now checks for the right ObsCore ID, though still recognises the wrong one (got from TAPRegExt), and warns if found.
- Fix TST input handler so TST files with fewer than 3 columns can be read.
- Add `Nd_cuboid` matcher option to match commands.

Version 2.5-3 (4 July 2014)

New and improved functionality:

- Add new command `cdsskymatch`. In most cases (for querying tables that can be found in VizieR) this can and should be used instead of `coneskymatch` - it's *much* faster.
- Commands `coneskymatch`, `sqlskymatch` and `pixfoot` will now guess RA/Dec columns if relevant parameters are left blank.
- Added new graphics output format `png-transp` to generate PNG files with transparent backgrounds.
- Upgraded Gregory Mantelet's ADQL library to version 1.2. Better ADQL parsing.

Improvements and adjustments to `taplint`:

- Rework `taplint` API to facilitate static acquisition of report codes during programmatic use. A few error codes have changed.
- Add new "duff query" test to `taplint`.
- Avoid `taplint` MDQ stage data type mismatch error report for BOOLEAN/boolean declared/returned data.
- `taplint` now takes steps to ensure that TAP_SCHEMA column list query is not truncated.
- `taplint` now flags absence of ObsCore table with I[nfo] not F[ailure] status.
- Change the implementation of `taplint` stages which perform validation against XSD schemas. Schemas from external namespaces may now be imported and used. The CPV stage, which was previously broken and disabled by default, is now fixed and enabled by default. Known/expected schemas are stored locally, and a warning is reported if external ones are used. Schema validation seems remarkably

complicated, so it's possible there are still errors in this implementation - if you suspect so, please report it.
- Add missing geometric reserved words to ADQL reserved word list. This fixes some problems with column names like "DISTANCE" in taplint tests.
- Fixed some bugs related to TAP table uploads. In particular these could cause incorrect table upload error reports in `taplint`.

Version 3.0 (3 October 2014)

New plotting commands:

A set of new plotting commands are provided which give comprehensive access to all the new-style visualisation capabilities available in TOPCAT v4. These commands are documented in Section 8. These commands, and the underlying visualisation facilities, are considerably more capable than the, now deprecated, old-style plot commands `plot2d`, `plot3d` and `plothist`.

Programmatic invocation:

Programmatic invocation of STILTS tasks from third-party java code is now officially sanctioned and documented in the new Section 12. To support this changes have been made to the parameter system (`Parameter` class now supports generics) and there are some visible changes to the user documentation as well (parameters now report their data type, and tasks report their classname). Normal (e.g. command-line) usage should not undergo any changes, but a fair bit of UI code has changed, so unexpected problems are possible.

Other items:

- Add new output mode `gui`, which displays the table data in a scrollable window on the screen.
- Add new `-allowunused` flag to the stilts command. If this is set, then unused parameter settings on the command line just result in a warning, not failure of the command.
- Attempting to write FITS tables with >999 columns now fails with a more helpful error message.
- Improved Unicode handling in VOTables. Fixed a serious bug in `votcopy` that generated unreadable output to BINARY or BINARY2 serialization if any non-empty column had datatype="unicodeChar". Also improved behaviour when copying between tables with unicodeChar columns; these are usually preserved now, rather than being squashed to datatype char. Some lurking Unicode-related issues remain.
- The TAP client now tolerates whitespace around UWS status codes.
- taplint: downgrade unknown post-table QUERY_STATUS value message from Error to Warning.

Version 3.0-1 (13 November 2014)

New functionality:

- Add (experimental) read-only support for Gaia/DPAC GBIN format.
- Add new task `tapskymatch`.
- Functions in class Coverage adjusted: new function `nearMoc`, and MOC can be identified by VizieR table IDs as well as by filename/URL.
- For `repeat` filter, add `-row|-table` flags to control sequence of output rows.
- For `setparam` and `repeat` filters, allow use of an algebraic expression for values, not just a literal value.
- Add special values `$ncol` and `$nrow` to the expression language to refer to the column and row counts in a table. The special variable `index` is also deprecated in

favour of `$index` or `$0`.

Bugfixes and minor improvements:

- Add some more colour maps for aux/density shading.
- Fix `stilts` invocation script to pick up classes from `stilts.jar` in script directory in preference to other places (e.g. topcat-full.jar).
- Fix `taplint` to permit application/xml not just text/xml content-type where appropriate (UWS stage).
- Fix `taplint` so it doesn't warn (W-TMV-UNSC) about unknown VOSITables schema.
- Fix `taplint` so that `unicodeChar` matches CHAR/VARCHAR in the same way as `char` for column type declaration purposes.
- Fix `taplint` so that capabilities document can have TAPRegExt dataModel ivo-id elements with xs:anyURI rather than vr:IdentifierURI (only a warning is issued in the latter case), in anticipation of TAPRegExt-1.0 Erratum #1.
- Adjust `taplint` to handle adql:TIMESTAMP columns more carefully on upload and retrieval.
- Update JSAMP to v1.3.5.

Version 3.0-2 (6 February 2015)

Plotting enhancements:

- Linear fitting of points is now available using the linearfit layer type for `plot2plane`. Points may be weighted.
- You can add titles to plots using the new `title` parameter.
- New plot layer type `sizexy` allows plotting (optionally autoscaled) markers with horizontal and vertical extents independently determined by input data.
- More flexibility when assigning colour maps, in aux and density shading modes, and spectrogram layer. New parameters `*func` allow assignment of different data->ramp mapping functions (sqrt and square as well as linear and logarithmic), and new parameters `*quant` allow quantisation of the colour map to discrete levels.
- Replace `maxsizeN` parameter with `autoscaleN` for `size` plot layer type. You can now optionally turn off autoscaling and specify marker size in pixels instead.
- Add `auxcrowd` parameter to plot2 tasks to influence tick crowding on aux axis colour ramp. Also adjust default to use fewer ticks.
- Add some "dart" options (fixed-base open or filled triangles) for plotting vectors (see `arrowN` parameter in layers like `xyvector`).
- Add some "triangle" options (variable-base open or filled triangles) for plotting ellipses (see `ellipseN` parameter in layers like `xyellipse`).
- Histogram normalisation option adjusted so that total area under bars, rather than total height of bars, is fixed.
- The `PlotDisplay` class that forms the result of plot2 commands can now have `PointSelectionListeners` registered on it. This lets you determine what point a user has clicked on if you're using the plotting classes from third party java code.

FITS I/O:

- Reworked part of the FITS table input implementation, in particular adjusting the way memory mapping is done to reduce resource requirements on some platforms. If you notice any difference, it should be reduced virtual and perhaps resident memory usage, and some (~10%?) performance improvements, when reading large FITS/colfits files. If you were previously having problems with large memory allocations leading to disk thrashing and system lockup when scanning files larger than RAM (this didn't happen on all OSes), these will hopefully have gone away. However, please report anything that appears to be working worse than before, or

continued memory usage issues.
- Colfits files can now be accessed from streams, not just uncompressed disk files (though that's not necessarily a good idea).

Bugfixes and workarounds:

- Fixed a query bug (missing `REQUEST=queryData` parameter) in the multi-SSA mode (`servicetype=ssa`) of `coneskymatch`. This long-standing bug would have stopped this command working at all with well-behaved SSA services.
- Fixed error in fits-var output (PCOUNT header card did not include block alignment gap).
- Graphics coordinates are now calculated in floating point rather than as integers. This fixes problems that could cause scaled vectors, ellipses etc to be drawn with shapes or orientations badly wrong due to rounding errors. It also improves plotting of analytic functions, especially to vector contexts (PDF/EPS).
- Fix some problems to do with zooming to very large/small plot axis ranges.
- Hide error bars (etc) that would extend to negative values on logarithmic axes; previously they were being drawn in anomalous places.
- Fix NullPointerException bug when null value was supplied to multi-word parameter (e.g. `tcube`).
- Fix Aux axis positioning for 3D plots so that the numeric labels don't get snipped off at top and bottom.
- Add a hack that allows LDAC FITS tables to be treated sensibly in auto-format-detection mode.
- Make VOTable handling more robust against unknown (illegal) datatypes.
- Add missing parameters `auxmax`, `auxmin` to plotting task documentation.

Version 3.0-3 (14 April 2015)

- New System Command option for input table syntax; you can now use "`<syscmd`" or "`syscmd|`" to supply input byte streams from Un*x pipelines.
- Add new Kernel Density Estimate plot layer types kde, knn and densogram for `plot2plane`.
- More histogram normalisation options provided. Instead of just `true`/`false`, the `normalisation` parameter of the histogram layer now has the options `none`, `height`, `area` and `maximum`. This allows both the area normalisation introduced in v3.0-2, and the height normalisation used in earlier versions which it replaced.
- More histogram bar style options provided; the histogram `barform` parameter now provides the options `semi_filled` (the new default) and `semi_steps`. These give outlined partially transparent bars, which make it much easier to see what's going on in multi-dataset histograms. Note `semi_steps` does not currently export very nicely to PDF/EPS. Similar options are also available in the new KDE plots.
- Column data read in as unsigned bytes will now be written out as unsigned bytes where the output format permits; previously they were forced to 16-bit signed integers. This affects FITS, VOTable and CDF I/O handlers.
- Add `count_rows()` method to JyStilts table objects, which for non-random tables may be much more efficient than `len()`.
- Be less strict about recognising colfits files (tolerate implicit TDIMn headers).
- `taplint` is now aware of, and performs some checks related to, schema-level table metadata declared by TAP services.
- Work round FITS read bug that could cause problems for VOTables using inline FITS serialization, and possibly elsewhere.
- Fix bug that caused trouble when auto-ranging a plot with a single sky position.

Version 3.0-4 (17 August 2015)

Bugfixes (some significant):

- Fix a serious bug in processing of FITS bit vector (`TFORMn='rX'`) columns. Values read from these columns are presented as a `boolean[]` array. In all previous versions of STIL the bits have appeared in that array in the wrong sequence (LSB..MSB per byte rather than the other way round). Apologies to anyone who may have got incorrect science results from this error in the past, and thanks to Paul Price for helping to diagnose it.
- Fix a less serious bug with `TFORMn='rX'` processing; attempting to read a single-element bit vector column (`TFORMn=1X` or `X`) previously resulted in an error making the file unreadable. Values read from such columns are now presented as Boolean scalars.
- Fix a VOTable reading bug relating to bit vector data (`datatype="bit"`) appearing in BINARY/BINARY2 serializations. This one was more obvious, it would usually generate an error when attempting to read the file.
- Fix serious bug in time conversion for CDF TIME_TT2000 data types.
- Fix a bug in `votcopy` that converted columns from datatype `unsignedByte` to `short` when transforming. Since v3.0-3 this is no longer necessary. In the case of converting to a binary serialization, since v3.0-3 this was causing it to generate unreadable VOTable output.
- Fix a bug in `votcopy` that failed to handle columns with datatype `bit`. In the case of converting to a binary serialization, these were in all previous versions generating unreadable VOTable output. Now they convert them to columns with datatype `boolean` (not perfect, but better).
- Fix `skyvector` bug: `dlat` and `dlon` values were being used the wrong way round.
- Upgrade JEL to v2.0.2. Fixes problem with evaluating void-typed expressions, and possibly some other obscure bugs.
- Some taplint bug fixes.

Behaviour changes:

- Changes to the way that TAP service table/column name reports are interpreted (to conform to original intention of TAP standard). Taplint now checks that table/column names from TAP_SCHEMA and /tables endpoint are regular-or-delimited-identifiers, but no longer submits example queries using supplied column names wrapped in additional quotes.
- Modify the heuristics that determine whether the first row of a CSV file is a header.

Enhancements (mostly minor):

- Added some functions to the Arrays class that return array-valued results from array-valued parameters: `add`, `subtract`, `multiply`, `divide`, `reciprocal`, `condition`.
- Improve error reporting in the face of non-VOTable TAP error responses. In many cases this makes it much easier to see what's going wrong with a TAP query.
- As a diagnostic tool, when making TAP queries, a log message giving a roughly equivalent `curl(1)` command is now issued at the CONFIG level (visible using flags `-verbose -verbose`).
- New `taplint` parameter `maxtable` limits the number of tables tested in the stage that queries data from each individual table (`MDQ`). May be useful for very large services.
- New `tapquery` parameter `upvotformat` to determine what VOTable serialization variant is used to transmit uploaded tables to the TAP server. Previously uploads were always BINARY which ought to work, but the parameter now defaults to TABLEDATA, since some services (e.g. CADC) currently fail with binary uploads.
- Minor improvement to version reporting (reports java specification version, no longer issues warning for absent revision string).
- Update JCDF library to v1.1 (minor changes to do with leap seconds).

Version 3.0-5 (22 October 2015)

- Fix error reporting bug when a non-VOTable response is received from a TAP service.
- Upgrade to JCDF v1.2 - fixes a bug when reading large (multi-2Gb) CDF files.
- Added source code for an example basic GUI plot application, `uk.ac.starlink.ttools.example.BasicPlotGui`.
- The expression language has a new way of referring to a column; if you use the form "`Object$` <*column-id*>" you get the value as an Object not a primitive. This is a special-interest measure for user-defined functions that need to see null numeric values.
- Adjust GBIN input handler: avoid descending into Class-typed members of gbin list objects, and add logging for object->column translations.

Version 3.0-6 (27 November 2015)

Crossmatching bug fix

Fix a long-standing crossmatch bug relating to range restriction during pre-processing. This could have caused missed associations (but not false positives) near the edge of coverage regions when using per-row errors, if the scale of the errors differed (especially differed significantly) between the matched tables. It affected `matcher` values of `<n>-d_err`, `skyerr`, `2d_ellipse` and `skyellipse` only. Thanks to Grant Kennedy (IoA) for reporting this bug.

Density plots

Some more options for making weighted density plots have been added. Since v3.0 the Density shading mode has let you see the density of plotted points, but this lacked some features. Three new ways to do density plots are added; these all give you the option of weighting by an additional coordinate (like the Aux mode), choosing the combination method (mean, median, sum, max, ...), and displaying the quantitative value-colour mapping on the shared colour ramp (previously aux axis) beside the plot. The new density plots are:

- Weighted shading mode, using shaped marker kernels on the screen pixel grid, available for all plot types
- SkyDensity layer, using HEALPix bins on the celestial sphere, for the Sky plot
- Density layer (*later obsoleted by Grid layer*), using square N*N screen pixel bins, for the Plane plot

The details are somewhat experimental and may undergo some adjustments in future releases (feedback welcome).

Colour maps

There are various changes affecting selection and display of colour maps used for density and aux axis shading:

- The default colour map for Aux mode, and other layers using the shared colour map, is no longer Rainbow! It's Inferno. Rainbow colour maps are much hated by visualisation experts. Of course you can still choose Rainbow if you like.
- Add some new colour maps: *Viridis*, *Inferno*, *Magma* and *Plasma* from Matplotlib 1.5, the *SRON* rainbow variant developed by Paul Tol, some diverging maps (*HotCold*, *RdBu*, *PiYG*, *BrBG*) and a qualitative constant chroma/luminance map *HueCL*.
- The options for Density and Aux shading are now mostly the same as each other except where there's good reason to differ. Previously they were different in haphazard ways.
- An attempt is made to give the default form of each colour map a sensible name, without leading minus signs.
- Fix it so that the whole range of each map is distinguishable from white. This is a good idea when you're plotting symbols on a white background, which is common in

stilts. Perhaps there are cases it's not such a good idea; if you think so, complain and I may change it back.
- Try to fix it so that all the colour maps go in the same direction (light->dark) where applicable.
- Throw out a couple of particularly useless colour maps.
- Colour map ramp display is now different for non-absolute maps; their effect is shown on a selection of base colours, not just for one base colour.

Minor items

- Try harder to identify epoch columns (suitable for time plot), in particular look for VOTable `xtype` of JD or MJD, and `units` of year.
- Add some functions to the Tilings class to do with solid angles (`healpixSqdeg`, `healpixSteradians`, `steradiansToSqdeg`, `sqdegToSteradians`, `SQDEG`).
- Fix plot bug; titles were painted in white for pixel output formats.
- Rationalise plot report logging. Some more diagnostic information about plots is now logged at the INFO level (visible if topcat is run with the `-verbose` flag).

Version 3.0-7 (10 June 2016)

Expression Language

The JEL library underlying the expression language parser has been upgraded to v2.1.1 (thanks to Konstantin Metlov), now supporting variable-length argument lists among other things. This allows the following improvements:

- New functions that support any number of arguments are provided: `array`, `intArray`, `stringArray` in class Arrays; `concat`, `join` in class Strings; and `sum`, `mean`, `variance`, `stdev`, `min`, `max`, `median`, `countTrue` in new class Lists.
- Some old lists of similarly-named functions with fixed numbers of arguments have been replaced by single functions that take an arbitrary number of arguments (e.g. `array(x1)`, `array(x1,x2)`, ... `array(x1,x2,x3,x4,x5,x6,x7,x8)` replaced by `array(values...)`).
- The 2-argument `min`/`max` functions in class `Arithmetic` have been renamed `minNaN`/`maxNaN` to avoid confusion, but in most cases existing expressions involving min/max will work as before.
- Some functions that used to require string arguments will now auto-convert numeric types (e.g. `concat(toString(RA),";",toString(DEC))` can now be written `concat(RA,";",DEC)`).
- You can now implement user-defined functions with variable numbers of arguments.
- Writing large (>=2**31) literal integers used to fail with an inscrutable error message. Now the message tells you to append the "`L`" character.

Time plots

The `plot2time` command has been enhanced so that it can make *multi-zone* plots - multiple plots stacked vertically that share the same horizontal (time) axis but have independent vertical axes. The time plot itself and this multi-zone feature are currently experimental; in future versions they may be improved or changed, and the multi-zone feature may be extended to other plot types. Some other changes and fixes have gone along with this:

- A few API changes have been made to support multi-zone plots, including generalising the `NavigationListener` interface and rearranging some `PlotDisplay` and `AbstractPlot2Task` constructor/factory method arguments. For single-zone plots the changes are not very substantial. This only affects you if you are using the STILTS classes as a java plotting library. If that applies to you and you have trouble upgrading, I'm happy to provide assistance.
- `plot2time` now supports shading modes (`shadingN` and associated parameters).

- The spectrogram layer now uses the (per-plot, or more precisely now per-zone) Aux colour map rather than a layer-specific colour map. This means that the colour ramp is displayed alongside the plot, but also that some parameters have been renamed (e.g. `auxmapZ` replaces `spectromapN`, where `Z` is an optional zone suffix and `N` is an optional layer suffix).
- The `function` layer type now works in the time plot, rather than throwing an error. However, it doesn't work very well, since the time coordinate is in unix seconds rather than something more user-friendly.
- Fixed a serious bug in ISO-8601 axis labelling. In some cases axis labels were being drawn at positions badly different from the correct position.

Miscellaneous enhancements and changes

- This and subsequent releases target **Java SE 6**, so will no longer run under the (now very ancient) Java 5 runtime.
- Provide more careful documentation of licensing arrangements. The distributed LICENSE.txt file notes that the starjava code is LGPL, and documents licenses for each third-party dependency.
- Add Fill layer type for Plane and Time plots.
- `tapquery` and `tapresume` now use blocking HTTP requests rather than repeated polls to wait for asynchronous TAP job completion from services that declare themselves UWS 1.1 compliant.
- Add new parameters `executionduration` and `destruction` to `tapquery` command. These let you request resource limit adjustments when submitting an asynchronous TAP job.
- Improve sky plot border painting.
- Clean up noisy Cubehelix colour map.
- New function `countTrue` in class Arrays.
- New stage `EXA` for taplint checks TAP `/examples` endpoint. Note the details of the examples format are still under discussion, (this version targets WD-DALI-1.1-20160415 & WD-TAP-1.1-20160428, somewhat informed by TAPNotes-2013-12-13), so the details may change in future.
- `taplint` now validates ObsCore 1.1 where declared alongside ObsCore 1.0. Currently uses PR-ObsCore-v1.1-20160330.
- The taplint API has changed slightly: the class that used to be `Reporter` is now called `TextOutputReporter`. If you are using taplint programmatically you may need to make small changes. This results from some refactoring that makes it easier to customise taplint output.
- Replaced `opaque` config option with `transparency` for plane and sky density plots.
- Changed implementation of GIF exporter for plots, from Acme to ImageIO. Shouldn't be any noticable difference. Acme encoding dependency removed.

Bug fixes

- Fix bug in cumulative histogram calculation.
- Fix read failure for FITS files with non-blank TDIM for zero-length columns.
- Fix bugs that led to timezone-dependent results when reading ISO-8601 or decimal year time columns.
- Fix numeric field truncation bug in LaTeX table output.
- Fix some parameter handling errors in `coneskymatch`.
- Fix NullPointerException bug for disjoint regions in some cases in `tmatchn`.

Version 3.0-8 (13 September 2016)

- New task tskymap lets you generate all-sky density maps (e.g. HEALPix) from an input table.

- New plot layer type healpix can plot all-sky maps from HEALPix map tables (e.g. as generated by `tskymap`).
- Add some HEALPix-related functions to the Tilings class: conversions from pixel index to sky position and conversions between ring and nested schemes.
- Subrange-typed plot command parameters like plot2plane's `auxclip` now have a default value of null. This uses a default clip, which avoids very light colours. Explicitly supplying a subrange value (e.g. "`0,1`") can now use the whole range; previously the very light colours were inaccessible.
- The GBIN input handler can now pick up more metadata from the classpath. For suitable tables, metadata included in datamodel classes if present can be interrogated to provide table and column descriptions and UCDs. There are still some deficiencies of this functionality (no column order, utypes and units missing, large file "temp.xml" written to current directory) dependent on issues in the upstream Gaia libraries and ICD.
- Add parameters `tablesurl`, `examplesurl` etc to task `taplint`, so you can specify custom locations for different TAP endpoints rather than have them fixed at their default locations hanging off the base service URL. Ditto for the tasks `tapquery` and `tapskymatch`, though in these cases the extra parameters are (since currently rather special interest) largely undocumented.
- Fix `taplint` behaviour in absence of a /tables endpoint (404 now gives a Warning not an Error).
- `taplint` ObsTAP validation updated to PR-ObsCore-v1.1-20160709.
- Fix bug that caused read failures for large (>0.5Gb) FITS files outside of the current directory on 32-bit JVMs. This was a regression bug since v3.0-2.
- Fix long-standing bug that failed to release file descriptors when opening FITS tables (could cause an error if very many FITS files were opened).

Version 3.0-9 (23 September 2016)

- Fix aux ranging bug in some plot types (SkyDensity, Healpix, Density, Weighted) that used an aux data range too small to show colour variations when `auxfunc=log` and negative data values were present.
- Improved performance (better memory use, faster) of some plot types (SkyDensity, Healpix, Density, Weighted).
- Fix JyStilts bug that could cause the same data to be used for different tables specified in a single plot.

Version 3.1 (8 March 2017)

Plotting improvements

- Improved documentation of plot layer types and shading modes in the user document - each option now has an example graphic and the text of the command that generates it.
- New layer type grid, to plot a 2-d weighted histogram. This replaces the `density` layer, which has been withdrawn (`grid` can do all the same things and more, except specify bin size in screen pixels).
- New plot layer type quantile, which can (e.g.) plot median lines through noisy data.
- New plot layer type gaussian, for Gaussian fits to histograms.
- Histogram-like layer types are now available from the plot2time command as well as from `plot2plane`, so you can now plot histograms with a temporal horizontal axis.
- New normalisation (scaling) option `unit` for histogram, KDE, and KNN plots.
- New normalisation (scaling) options `per_day` etc for histogram, KDE and KNN layers when used from the `plot2time` command.
- Colour names recognised by the `plot2*` command `*color` parameters now include the (140) CSS-like colours alongside the dozen standard plotting colours.
- Plot commands `insets` parameter now lets you specify margins round plots using any combination of `<top>,<left>,<bottom>,<right>`, rather than requiring all

values or none.
- Add new parameter `auxwidth` to plot2 commands, to control colour ramp lateral dimension in pixels.
- Modify *plan* caching arrangements for STILTS plots. This results in much better interactive performance (navigation etc) for some plot layer types plotted to the screen. There are also some minor API changes to a few of the plot2 utility classes, which now provide more flexibility for caching.
- Various tweaks to the details of how plots are positioned on the screen or in output graphics files.

Miscellaneous enhancements

- Fixed the match score (distance measure) for combined matchers. Previously, the score was a linear sum of the unscaled distances for the constituent matchers, which meant a *Best* match was pretty meaningless. Now, it adds scaled distances in quadrature, so Best matching should give you a somewhat sensible result.
- The `skyerr` and `nd_err` matchers now report output separations as scaled (dimensionless) values rather than in physical units; this means the results are more comparable, so *Best* matches will make more sense.
- taplint can now optionally write its output in JSON format (see `format` parameter).
- Update `taplint` OBS stage for PR-ObsCore-v1.1-20160923.
- The Maths function `hypot` now takes an arbitrary number of arguments (instead of exactly two).
- Add `blockmaxrec` parameter to tapskymatch, and improve warning text in case of result truncations.
- Improve `tapquery`/`tapresume` behaviour when monitoring async jobs; if the service goes down, the application will wait for it to come back rather than bailing out.
- Options for MOC output format (`mocfmt` parameter in pixfoot command) have changed; option `ascii` is replaced by `json`. The old ASCII format was slightly broken JSON in any case.
- MOC library upgraded to v4.6 (from v3.3). Improved MOC output.
- Update JCDF library to v1.2-2 (2017-01-01 leap second).

Bug fixes

- Fix KDE/KNN plotting bug that could get scaling badly wrong for normalised cumulative plots.
- Fix some regression bugs relating to `plot2` command `insets` parameter, present since v3.0-6.
- Remove spurious padding from EPS graphics output.
- Fix small bug in `plot2plane`/`plot2time` label painting; horizontal axis label was sometimes off the bottom of the plot by a few pixels.
- Fix subpixel offset of colour ramp frame in PDF/PostScript graphics output.

Version 3.1-1 (29 September 2017)

New functionality

- New plot layer types xycorr and skycorr for error ellipses rotated as specified by Gaia-style correlation values.
- New plot layer type skygrid can draw multiple sky system coordinate axis grids on sky plot.
- Colour map parameters (`auxmap`, `densemapN` etc) for the plotting commands will now accept custom colour maps that interpolate between a list of named colours, e.g. "`HotPink-yellow-SkyBlue`".
- New position angle calculation functions `posAngRadians` and `posAngDegrees` added to expression language.

Table I/O changes

- It is now possible to write and re-read tables with >999 columns to FITS or colfits format. The new limit is 2^31 columns. This uses a non-standard convention; software that is not aware of the convention (e.g. CFITSIO or earlier STILTS versions) will only be able to use the first 998 columns of tables written in this way.
- For VOTable columns that reference `COOSYS` elements, the relevant information is now accessible as column auxiliary metadata (`CoosysSystem`, `CoosysEpoch`, `CoosysEquinox`) e.g. using the `meta` filter.
- Any columns referencing `COOSYS` elements read from VOTable-based formats (VOTable or FITS-plus) will now be written out to VOTable-based formats with equivalent COOSYS references included. Currently not table parameters though.
- The default version for output VOTables is now VOTable 1.3. New output formats `votable-binary2-inline` and `votable-binary2-href` are now offered alongside the five previously available VOTable variants.
- Slight changes to the FITS-plus output handler VOTable formatting in the primary HDU; now uses default output VOTable version rather than VOTable 1.1.
- FITS keywords using the ESO HIERARCH convention can now be read as table parameters rather than ignored when reading FITS tables.

Minor behaviour changes

- The `charset` parameter of `votcopy` now defaults to UTF-8 rather than the system default.
- Replace parameter `autoscale` with `unit` in `skyvector` and `skyellipse` plot layers.
- Change autoscaling defaults for plot layer types `xyvector`, `xyzvector`, `skyvector`, `xyellipse`, `skyellipse`; autoscaling is now turned *off* by default. Apologies for this change to behaviour, but it's better for consistency with new layer types `xycorr` and `skycorr`, and presents less danger of misinterpreted plots.
- Taplint: downgrade type mismatch error `E_QTYP` to warning `W_QTYP` in view of TAP-1.0 Erratum #3.
- The `COOSYS` element is now passed without a deprecation warning in VOTable 1.3 documents by the `taplint` and `votlint` validators, in view of VOTable-1.3 Erratum #1. The warning is still issued for VOTable 1.2 documents, but the text is toned down.
- Taplint no longer issues a warning about DataModel references that do not match the `vr:IdentifierURI` type; this follows TAPRegExt Erratum #1.
- Modify taplint Examples document @vocab attribute testing in the light of DALI 1.1.

Bug fixes

- Update PixTools (HEALPix) library to 2017-09-06 version (https://github.com/kuropat/eag-HEALPix, `447a7be073876dba32`). This fixes a bug in `healpixRingIndex` that could give the wrong value for small values of longitude near zero in the equatorial region. It seems possible that this might have led to very infrequent missed associations when crossmatching in these regions, but tests appear to indicate that no such errors would actually have resulted.
- Long fields (>10240 characters) in output CSV files are no longer truncated.
- Fix misfeature in skyvector and skyellipse plot layers; these now preserve orientation on the sky even when `viewsys` differs from `datasys`. Previously the `dlat`/`dlon`/`posang` parameters were always interpreted in the view, rather than the data, sky coordinate system.
- Fix bug in plot title placement.
- Fix filename generation error for plotting command animation output (wrong number of digits if frame count was an exact power of 10).

Version 3.1-2 (7 November 2017)

New expression language functions

- New functions `desigTo*` for extracting positions from IAU-style object designations (like `2MASS J04355524+1630331`) - use with care.
- New functions `polarDistanceDegrees` and `polarDistanceRadians`; these calculate the distance in 3d space between two positions specified in spherical polar coordinates.
- New functions `phase` to help with phase folding given a known period. A new modulus function `mod`, whose output is always positive (unlike the `%` operator) is also added.

Bug fixes and workarounds

- Upgrade JEL to v2.1.2: now you can use functions in the expression language that have the same name as table columns.
- Add fix to discard on read any VOTable/FITS-plus table parameters with names "`uk.ac.starlink.topcat.plot2.TopcatLayer*`". These useless items were added in potentially large numbers when saving plotted tables from TOPCAT v4.5 (TOPCAT bug). A null `tpipe` operation at this version will therefore purge these items.
- Fix plot bug that sometimes caused error bars to come out very small when exporting to a graphics file. This was a regression bug (present in v3.1-1 but not v3.1).
- Fix another tiny error bar plotting bug too.
- Small taplint update; required UCD for `obs_publisher_did`/`publisher_id` changed to match REC-ObsCore-1.1 (even though it's an illegal UCD1+).

Version 3.1-3 (24 April 2018)

New Functionality

- New DataLink validator task `datalinklint`.
- Add class Gaia to the expression language. This contains functions for estimating distances from parallaxes and propagating astrometric parameters and errors to different epochs. The functions are not specific to data from the Gaia astrometry satellite, but are presented in a form convenient for use with the Gaia DR2 source catalogue.
- New array manipulation functions `slice` and `pick` added to class Arrays.
- New utility function `square` added to class Maths.
- The `addcol` and `colmeta` filters now accept `-shape` and `-elsize` flags for defining array and string shape/extent column metadata.
- DataLink-style service descriptor RESOURCEs in input VOTables (and FITS-plus tables) now appear as table parameters.

Plotting Enhancements

- Various improvements to the Contour plotter. You can now contour quantities weighted by a given coordinate, rather than just point density. The smoothing from weighted point samples to the grid being contoured can be performed using sum, mean and other combination methods, it now uses a Gaussian kernel rather than a square top hat, and performance is considerably improved especially at large smoothing widths. The contour levels used are now *reported*, so can be seen by specifying the `-verbose` flag. The contours are now plotted correctly right up to the edge of the visible plot, and various bugs have been fixed.

- The histogram plotter now has a `combine` parameter (`sum`, `mean`, `median`, `min`, `max`, `stdev`, etc) for weighted histograms. This means that you can plot e.g. the mean value of a given quantity per interval on the X axis rather than just the summed quantity. A corresponding (though somewhat less well-defined) option is also provided for the kde and densogram plotters.
- The `combine` parameter that configures how values are binned in various histogram-like plots (skydensity, healpix) grid, and since this version also contour, histogram, kde) now has two new options, `sum-per-unit` and `count-per-unit`. These work like the existing `sum` and `count` options, but scale the combined values by the relevant unit (e.g. X axis unit or solid angle). Where these units are physical, a `perunit` parameter is also provided for scaling in convenient units: second, day, year etc for time (plot2time), and square degree, arcminute, arcecond etc for solid angle (skydensity, healpix). In some cases, the `combine` default values have changed.
- The same new `combine`/`perunit` options are also added to the tskymap command.
- The linearfit layer type is now available for `plot2time` as well as `plot2plane`.

Minor behaviour changes

- `votlint` no longer considers an integer- or array-typed `PARAM` element with attribute `value=""` to be an error.
- `votlint` no longer Warns about `PARAM/@ref->FIELD/@ID` references, since this usage pattern is now found in DataLink Service Descriptors.
- The IPAC table reader now matches data type specifications case-insensitively.
- Improvements in documentation of the expression language functions: in the Function documentation section classes are now listed in alphabetical order, and examples are included in some cases. Some readability improvements have also been made in the function browser displayed by the `funcs` command.

Bugfixes

- `taplint`'s VOTable validation now includes XML schema/DTD validation as well as the custom VOTable checks. It was always supposed to do this, but it seems that it didn't, at least in recent versions.
- Update JCDF to v1.2-3; fixes some CDF reading bugs.
- Prevent `coneskymatch` from sending cone search queries with `sr=NaN`.
- Minor fix in `taplint` OBS stage; ObsCore v1.1 `s_region` UCD is `pos.outline;obs.field`, not `phys.outline;obs.field` (a relic from PR-ObsCore-1.1-20161004).

Version 3.1-4 (18 May 2018)

Bugfixes and minor enhancements:

- Add new option `delete=now` to `tapresume`, which lets you delete a TAP async job. Also add a message to `tapquery` inviting the user to delete interrupted jobs in this way.
- Upgrade Grégory Mantelet's ADQL parsing library to v1.4.
- VOTable output tweaked: single quotes now used for attribute values if attribute contains double quotes.
- Fix bug in `tapquery` and `tapresume` that caused a result read failure for result VOTables longer than 2Gb.
- Fix range bug; caused plot failure when plotting very large values with no variation (e.g. gaia_source solution_id).

Version 3.1-5 (2 November 2018)

Plotting:

- New spheregrid plot layer, for plotting a spherical net around the origin, added to

`plot2cube` and `plot2sphere`.
- New line3d plot layer, for plotting lines joining points in 3d, added to `plot2cube` and `plot2sphere`.
- New `sortaxis` parameter, for joining out-of-sequence points, added to line plot layer.
- Replace skygrid plot layer parameter `crowd` with two parameters, `loncrowd` and `latcrowd`, so you can control grid line spacing for meridians and parallels independently. Note this is a backwardly-incompatible change, so could break existing scripts. This change may have introduced slight changes to sky axis grid line spacing at low crowding levels.
- Contour level calculations improved; in some cases this previously didn't work well at low point density, resulting in missing contours. The meaning of the `zero` parameter has also changed slightly, it now defaults to 1 and is not phase folded.
- Sphere plot now centers axes on zero if the center would otherwise be near zero.
- Address long-standing plot auto-ranging issue; when auto-ranging resulted in the bottom X limit or right-hand Y limit being exactly equal to zero, the corresponding zero-valued results were not plotted. Auto-ranging has been slightly adjusted to avoid that. This results in pixel-level changes to plot appearance in some cases; these can be avoided by explicitly setting `xmax`/`ymin` etc.

Expression Language:

- New functions in class Gaia: `polarXYZ`, `astromXYZ`, `astromUVW`, `icrsToGal`, `galToIcrs`, `icrsToEcl`, `eclToIcrs`. These can calculate Cartesian position and velocity components from (e.g. Gaia) astrometric parameters.
- New Shapes class added for working with polygons in the X,Y plane; has functions `isInside` and `polyLine`.
- Rearrange expression language documentation slightly: there is a new section listing Special Tokens. This includes "`$random`", which was previously undocumented and named "`RANDOM`".
- New functions `urlEncode` and `urlDecode` added to class Strings.
- New convenience function `exp10` in class Maths.

Validators:

- Improvements to `taplint` UWS stage: now validates job documents against UWS schema, tests `uws:job/@version` attribute, and does improved and version-sensitive (UWS v1.0/v1.1) validation of UWS timestamps. Also does more complete deletion of submitted jobs.
- `taplint` schema validation now takes account of VOTable 1.3 Erratum #2 - `FIELD/@precision` attribute values "`F0`"/"`E0`" are now permitted.
- The VOTable validator in `votlint` and `taplint` no longer complains about multiple `INFO` elements in the same scope sharing the same `name` attribute.
- Minor improvements to `datalinklint` error reporting.

Miscellaneous Enhancements:

- New `collapsecols` filter added, to convert a run of adjacent scalar columns into an array column (the opposite of `explodecols`).

Minor enhancements and behaviour changes:

- GBIN read fix to work around changed behaviour in recent GaiaTools (versions 19.4.*, >=20.1.0 and >=21.0.0) that caused GBIN table reading to fail.
- Fix some additional commands (`votcopy`, `votlint`, `datalinklint`) so they follow HTTP 3xx redirects automatically in more cases when given a URL for input. Most STILTS commands already do this.
- Introduce a couple of measures to reduce the likelihood of unintentional service

overload from `coneskymatch` and friends: add a progressively increasing delay for `erract=retry*` error handling modes, and decrease the soft maximum for the `parallel` parameter to 5 (from 10).
- Slight improvements to the JDBC Configuration section of this manual.
- Small change to MOC handling that might possibly avoid some network-related performance issues.

Bugfixes:

- Fix `function` plot layer so that NaN values are omitted rather than interpolated.
- The `stilts` startup script now correctly follows symlinks on some OSes where it didn't work before (including OSX). It also looks for jar files to use in slightly different places on OS X.

Version 3.1-6 (9 May 2019)

Plotting enhancements

- Introduced new colour scaling options `histogram` and `histolog` for the parameters `auxfunc`, `densefunc` etc in `plot2plane` and the other plotting commands. This can make it much easier to see structure in quantities that do not vary smoothly over their min-max range.
- New plot layer types poly4, mark4, polygon introduced for drawing outline or filled quadrilaterals and other polygons in all plot types.
- The `line` plot layer now has an `aux` parameter that can vary the colour of the line along its length according to some third quantity.
- `plot2sky` now by default draws a small scale bar at the bottom left corner of the plot, indicating the scale in degrees, minutes or seconds. It can be switched off using the new `scalebar` parameter.
- New option `Car0` for the `plot2sky projection` parameter; this is like `Car` (Plate Carrée) but has longitude=0 at the left/right edge rather than the center of the plot.
- The healpix plot layer can now plot HEALPix levels up to 20 (previously the maximum was 13). Also, autoranging now works, so if a plot is made of a HEALPix file covering part of the sky, the sky view is centered on it in the same way as for marker plots. Memory management is improved for fairly large maps.

HEALPix-FITS support

Various changes introduced to support the semi-standard HEALPix-FITS serialization convention. Available information about HEALPix encoding (level, index column, ordering scheme, coord sys) can now be stored in custom table parameters (of the form `STIL_HPX_*`), and is used by FITS output handlers to insert the relevant FITS headers. The existing FITS handlers do this where it's not disruptive, and the new fits-healpix output handler will additionally move and rename columns if required. This metadata is round-tripped by FITS and VOTable I/O handlers. It is added automatically by the tskymap command, and can be manipulated by hand using the new healpixmeta filter. FITS support is not perfect: the `BAD_DATA` FITS keyword is ignored, and the 1024-element array-valued column variant is not understood.

VOTable 1.4 support

Support has been introduced for version 1.4 of the VOTable format and its new `TIMESYS` element.

- `votlint` (and hence `taplint`) now supports Version 1.4 VOTables: the 1.4 schema is used for XSD validation, and the TIMESYS element is checked for attribute content and suitable referencing.
- For VOTable columns that reference TIMESYS elements, the relevant information is now accessible as column auxiliary metadata (`TimesysTimeorigin`,

`TimesysTimescale`, `TimesysRefposition`) e.g. using the `meta` filter.

- Any columns referencing TIMESYS elements read from VOTable-based formats (VOTable or FITS-plus) can now be written out to VOTable-based formats with equivalent TIMESYS references included, so TIMESYS round-tripping for columns works; however this will only be done if the VOTable output format is set to version 1.4. By default (at least as long as 1.4 is not finalised) the output version is 1.3. To enable this TIMESYS output, set the system property `-Dvotable.version=1.4`. Currently this TIMESYS output works only for table columns (FIELDs) not parameters (PARAMs).
- The `timeoffset` attribute of a TIMESYS element referenced by a VOTable column is used to make sense of column data when interpreting it as an absolute time. Currently, the only use of this is in time plots.

This VOTable 1.4 support has resulted in some minor related behaviour changes:

- `FIELD/@ref` attributes are no longer imported as "`VOTable ref`" column aux metadata items, since they often interfere with TIMESYS references. Doing this was probably always a bad idea since the referencing is not kept track of within the application, so withdrawing this functionality makes sense, but beware that it might change or break some existing behaviour.
- `votlint` modifications above may now interrogate external vocabulary resources during validation, meaning that external network connections may be made during validation, which didn't happen in previous versions.

Other enhancements

- New functions `indexOf` in class Arrays to find position of a given value in an array.
- New functions `parseDoubles` and `parseInts` in class Conversions for extracting array values from strings (experimental).
- Fix expression compilation so that `$ID` column references referring to nonexistent columns are rejected at compile time rather than causing trouble during evaluation.
- The validator tasks `taplint` and `datalinklint` now insert the token "`(IVOA-validate)`" into the `User-Agent` header of all HTTP requests they make, for convenience of services that want to identify validators. This is a convention discussed within the IVOA Operations IG.

Bug and misfeature fixes

- Improve colour ramp quantisation (plot parameters `auxquant` etc); the full color range is now included.
- Use period not comma as decimal separator for non-sexagesimal `plot2sky` axis labels regardless of Locale; also avoid trailing comma sometimes erroneously present.
- Fix bug/misfeature in CDF table parameter construction: CDF global attributes were ignored (with a "WARNING: Omitting complicated global attribute" message) if they contained any null entries. Now such entries are just ignored and the table parameter is constructed from the global attribute using the non-null entries.
- Fix some bugs relating to plotting values close to the limits of the double precision range.
- Fix a problem with the link2 plot layers on sky plots with Aitoff/Car projections that caused short lines that should span the antimeridian to appear as long lines crossing the whole sky. Such links are now just not drawn.
- Fix misleading error message about `seq` parameter for underconfigured plot layers in `plot2*` commands.

Version 3.2 (18 November 2019)

Runtime environment: Java 8

From this release, **STILTS requires Java 8** (a.k.a. Java 1.8) or greater to run, rather than Java 6 as for previous releases. Java 8 has been around since 2014, so it should be available on all but very ancient platforms. If execution fails with a `java.lang.UnsupportedClassVersionError` then you need to upgrade.

New or enhanced functionality:

- New `cone` command to execute simple spatial DAL queries (Cone Search, SIA, SSA).
- Most plot layer types, though not the intial data preparation, will now run in parallel for large datasets. When using one of the `plot2*` commands interactively (`omode=swing`), this should make interacting with slow plots faster on multi-core machines. (In rare cases this multi-threading might cause problems with memory usage; it can be effectively turned off if required by using the system property `java.util.concurrent.ForkJoinPool.common.parallelism`.)
- New class URLs contains expression language utility functions for constructing certain service URLs: `hips2fitsUrl`, `bibcodeUrl`, `doiUrl`, `arxivUrl`, `simbadUrl` and `nedUrl`. Existing functions `urlEncode` and `urlDecode` have been moved to `URLs` from class Strings.
- New functions `parseBigInteger` and `parseBigDecimal` in class Conversions.
- Add new parameter `usepos` to `polygon` plot layer, to toggle inclusion of reference position in polygon vertex list.
- Add parameter `interface` added to TAP query tasks `tapquery`, `tapskymatch` and `taplint`. This can control how TAP version is determined.
- Modified behaviour for offset (e.g. unsigned) longs in FITS files. 64-bit integer columns (`TFORMn='K'`) with non-zero integer offsets (`TSCALn=1`, `TZEROn<>0`) are now represented internally as Strings; previously they were represented as Long integers, but values out of the possible range appeared as null (with a warning written through the logging system). Such columns are most commonly seen representing unsigned long values. If written back out to FITS, the offset long value will be reinstated, but other output formats cannot encode unsigned longs, so they will stay as strings.

Minor enhancements and behaviour changes:

- VOTable output now writes VOTable version 1.4 by default (was 1.3).
- Modified plot legend display so that small markers are displayed a bit bigger in the legend than on the plot for readability.
- Add new colour map Cividis.
- Permit FITS and VOTable files with zero-length string columns. Previously all-null or zero-length string columns were sometimes forced to single-character values.
- Update mapped file unmapping implementation to work for java9+.
- Updated `taplint` OBS stage according to ObsCore-1.1 Erratum #1 (corrected UCDs). No code changes required for ObsCore-1.1 Erratum #2.
- More taplint tests added for TAP 1.1-style capabilities document.
- Taplint now recognises ADQL 2.1 language features (as listed in PR-ADQL-2.1-20180112), avoiding some `E_KEYX` errors.
- VOResource validation in `taplint` now uses VOResource v1.1 not v1.0. Erratum #1 (multiple security methods) has been applied.
- Some `taplint` enhancements: add some new general tests, and some only used when testing TAP 1.1 services.
- Adopt VOTable-1.3 Erratum #3 in `votlint`; `arraysize="1"` now provokes a Warning.
- Updates to `meta` filter documentation.
- Minor changes to behaviour when querying TAP 1.1 services (REQUEST parameter is omitted).
- JSAMP to version 1.3.7.

Bugfixes:

- Fix regression issues introduced at v3.1-6 relating to `pixsample` command; it now reads HEALPix metadata from headers correctly again.
- Avoid sometimes losing precision when reading ASCII/CSV values in the range +/-(1e-45..1e-38).
- Fix plot axis ranging bug: padding was not always applied properly for logarithmic axes.
- JEL bug fix update, to avoid unwanted debugging output for String function null returns.

Version 3.2-1 (5 June 2020)

File Formats:

- The **ECSV** (Enhanced Character Separated Values) storage format is now supported for input and output.
- The **Feather** storage format is now supported for input and output.

Performance:

A number of implementation changes have been made which may improve performance, particularly for crossmatching (typical improvements for large sky crossmatches are a factor of 2, though YMMV). These should have no effect on the results, but if anybody notices crossmatching behaviour which is changed since previous versions or otherwise suspicious, please report it.

- The HEALPix implementation has been replaced; all HEALPix manipulation is now done using the excellent cds-healpix-java library written by François-Xavier Pineau from CDS, which speeds up sky crossmatching considerably. Many thanks to François-Xavier and to CDS for providing this library and for assistance with its use; thanks also to Nikolay Kuropatkin from FermiLab whose PixTools library served this purpose in STILTS up till now.
- Rows are now binned during crossmatches using a HashSet rather than a TreeSet.
- Evaluations of the `arcsin` function in Sky matches now use the (Apache via cds-healpix) FastMath implementation rather than the standard J2SE version.
- Performance is improved when reading long String values from FITS files.

Plot commands:

- The new layer types `area`, `central` and `arealabel` can be used for plotting region data supplied as area coordinates in the form of STC-S (e.g. from ObsCore/EPN-TAP `s_region`), DALI polygon/circle/point, or (ASCII) MOC columns. These layers are available from the `plot2plane`, `plot2sky` and `plot2sphere` commands.
- The `plot2time` command has been improved, and is no longer considered *experimental*. A new `ttypeN` parameter is added, which allows you to explicitly define (as MJD, JD, ISO-8601 etc) how input values are mapped to time. If no such mapping is specified or can be guessed from the input metadata, time values are now interpreted by default as MJD rather than (as previously) Unix seconds.
- Provide more options for the **shapeN** parameter of the plot `mark` layer type: `fat_circle`, `fat_cross` etc using thicker lines.
- Add line thickness parameter `thick` to `contour` plot.
- Plot `storage` parameter has new options `disk`, `policy` and `persistent`. These allow off-heap plot data caching, so that interactive plots for very large data sets may be made without running of of memory. The `persistent` option additionally means that results of expensive data read operations can be cached between invocations.
- The `line` layer's `sortaxisN` parameter now accepts the option `time` rather than `X` (or

Y) when used with plot2time.
- Improved plot axis labelling in LaTeX mode, e.g. in LaTeX write "3x10^6" not "3e6".
- Add new option anchorN=center to the label plot layer.
- Add parameters minsize and minshape to poly4 plot layer type.
- Improve accuracy when drawing large HEALPix tile boundaries in some cases for skydensity and healpix sky plot layers.

Other minor enhancements:

- Basic support for SIA version 2 as well as version 1 in cone and coneskymatch commands (new option servicetype=sia2).
- Add new conversion functions *ToUnixSec to Times class.
- Add new functions midLon and midLat to Sky class.
- Slightly improve sampling accuracy of pixsample output.

Bug and misfeature fixes:

- Fix bug in SkyGrid plot layer that made it ignore the viewsys parameter, always assuming equatorial so plotting the wrong grid lines for non-equatorial view sky systems. Note this bug applied only to STILTS, not TOPCAT.
- Fix regression bug since last release that refused to make dataless plots.
- Fix polygon plots in plot2sphere - this combination was failing to plot anything, now it works.
- FITS ASCII table extensions with TFORM values of In are now treated as 64-bit integers for n>=10 rather than n>10.
- Modify taplint processing of QUERY_STATUS declarations; overflow may now be declared before, as well as after, the table.
- Slightly improve plot axis labelling for small numbers; remove unnecessary decimal point in scientific notation in some cases.

Version 3.2-2 (24 August 2020)

New Functionality:

- SVG (Scalable Vector Graphics) is now one of the supported graphic output formats for plot export.

Bugfixes and minor enhancements:

- Fix Healpix trouble at large angles, e.g. sky crossmatch failure with match radius >6 degrees.
- Fix ECSV output bug: encoding was incorrect for metadata scalars with certain non-alphanumeric first characters, leading to invalid YAML.
- Votlint now warns about use of vocabulary terms marked as *preliminary* or *deprecated* as well as unknown ones (although currently no such terms exist in relevant vocabularies).
- Remove some unhelpful per-column metadata items from ECSV output.
- Improve seeding of the expression language $random special value; it should now be less dependent on JVM details.
- TAP curl(1) equivalent logging flag error fixed (write --compressed not --compress).

Version 3.3 (23 October 2020)

Plot server

A plot server has been added that can serve interactive plots in web applications for server-side tables. Documentation for the server command has been mostly moved into

new Server Mode section.

Bug fixes

- Upgrade cds-healpix-java library to v0.28_1; avoid occasional sky crossmatch failures (termination with error).
- Fix minor plotting bug that could cause white points to be invisible.

Version 3.4 (11 January 2021)

In this version, the table handling library STIL has been upgraded to v4.0, which enables some enhancements including multithreading and new I/O handler features.

Notable new functionality:

- Auto file format detection now examines filenames to help guess format; this means that e.g. for CSV files named with the "`.csv`" extension, it is no longer necessary to specify `ifmt=csv`.
- Several operations can now run in parallel, so should execute faster for large datasets on multi-core machines. Parallelised operations are the output modes stats and count, the filters stats and sort, and the task tskymap. The `stats` and `sort` filters have `-[no]parallel` flags that can be used to select between sequential and parallel execution. Parallel data caching for plots is also available using the `storage=parallel` option, but this is currently experimental and not generally recommended. Future releases will parallelise other operations, including matching.
- I/O handlers can now in some cases take parenthesised options, e.g. `ofmt=votable(version=V12,format=BINARY)`. For examples see individual handler documentation, e.g. the VOTable output handler and ECSV input handler.
- Scheme specifiers can now be used to specify input tables, to read tables not based on external files. Options currently provided include simple sequence, simulated sky and strange attractor data.
- Improved documentation of I/O formats in this document: full docs are now provided in Section 5.1 for all formats rather than summaries with pointers to SUN/253. The Table I/O section has been somewhat reorganised and tidied up.

Minor enhancements:

- When the stats filter calculates quantiles, the `-qapprox` flag can be supplied to use an (approximate) algorithm that will not run out of memory.
- Add `combine` parameter to `tcube`, so it can produce generalised weighted maps.
- Add new options `acos` and `cos` for colour scaling options `auxfunc`, `densefunc` etc; these provide linear-like stretch functions with steeper/flatter ends, which may be useful for shading by quantities with most variation near to/far from the middle of the range.
- The progress filter no longer slows down pipelines to any noticeable extent.
- COOSYS and TIMESYS attributes are now preserved during VOTable I/O for table PARAMs (as well as for FIELDs, which was already the case).
- Tidy up open files in the table concatenation command `tcat`; with `lazy=true` it should no longer fail with "Too many open files".
- The every filter has a new `-exact|-approx` flag; `-approx` may be required for certain operations to run in parallel.
- Slightly improved documentation for `cdsskymatch`.

Bugfixes:

- Fix bug that could give unhelpful table load error message for very short non-FITS files.
- Fix broken example in `tmatchn` documentation.
- Fix `taplint` to cope with VOSI1.1 `detail=min`-style response from `/tables`

endpoint; avoid spurious E-TMC-CM21/E-TMC-FM21 errors.
- Cope better with infinite values in aux plot coordinates.
- Fix histogram ranging bug (plot failure under certain circumstances).
- Upgrade JEL to v2.1.2: bugfix plus some efficiency improvements in object creation.
- Avoid inserting NULL_VALUE custom metadata entries into ECSV output.
- Make taplint changes for later edits to ObsCore-1.1 Erratum #1 (dataproduct_type and dataproduct_subtype UCDs are now meta.code.class).

Version 3.4-1 (10 June 2021)

File formats:

- Apache Parquet format is now supported for input and output (note not available in all configurations).
- AAS Machine-Readable Table (MRT) format is now supported for input.
- ECSV format input and output handlers are upgraded to version 1.0 of the ECSV format, meaning they can now read and write array-valued columns.
- Add configuration option header for CSV input handler, to indicate whether header line is present.
- Add configuration option maxSample for CSV and ASCII input handlers to reduce 2-pass read time.
- Variable-length array-valued columns in FITS tables (P/Q descriptors) can now be read even in compressed or streamed input.

Other new functionality:

- Add new plot layers to plot2plane for plotting data (such as spectra or time series) in array-valued columns: lines, marks, yerrors and xyerrors.
- Add new functions to class Arrays for working with array values: arrayFunc, intArrayFunc, sequence, constant.
- Add new table scheme test.
- The cumulative option in histogram-like plots can now take the values none/forward/reverse, not just true/false.
- Introduce the aitoff0 option alongside aitoff as a value for the plot2sky projection parameter. In the same way as car/car0, this gives an Aitoff projection but with the longitude=0 line at the left/right edge instead of the center.
- Modify tcube so that it accepts expression language expressions rather than just column name/ID values for the cols and scale parameters.

Taplint changes:

- Add new taplint stage LOC, for ObsLocTAP validation. This release validates against the draft standard version PR-ObsLocTAP-20210609.
- Taplint stage TMV now validates against (a pre-REC version of) version 1.2 of the VODataService schema instead of version 1.1.
- Downgrade taplint report for accepting unknown LANG from E_DSUC to W_OLNG.
- Relax taplint timestamp upload round-tripping requirements; folding adql:TIMESTAMP->timestamp no longer provokes error E-UPL-TMCX.
- Relax some taplint type checking for TAP_SCHEMA columns of TAP 1.0 services (see TAP 1.0 Erratum #3); report E-TMS-TSCT is downgraded to W-TMS-TSCT.
- Taplint W_EXVC TOPCAT compatibility warnings are downgraded to I_EXVT.

Minor enhancements and workarounds:

- The fits-var output handler now avoids use of the THEAP keyword (no pre-heap gap is written). Heap padding is legal FITS, but bugs in other FITS software mean that

some third party components (including `fverify` in FTOOLS e.g. v3.14-3.50) have problems with such files.
- Fix server to work with scheme-specified tables having a non-default `tablefactory` parameter.
- JDBC output no longer attempts to create VARCHAR(0) columns.
- Output of `cache` filter now retains no reference to input table, so it can be used to manage deeply nested tables in (Jy)STILTS pipelines.
- JyStilts now has access to the same table schemes as STILTS.
- Collapse whitespace in some table metadata description items in output form `meta` filter and output mode. This prevents unwanted linebreaks as well as tidying whitespace-formatted values.
- Space-delimited ECSV files now write empty fields quoted.
- Unknown or unsupported column datatype values in ECSV files are now treated like `string` rather than causing table read failure.
- Empty strings in FITS 1-character columns are now returned as blank values rather than ASCII NUL ('`\0`').
- Undersized, including zero-length, strings written to FITS columns are now by default terminated with an ASCII NUL rather than in some cases padded with spaces.

Bugfixes:

- Fix regression bug in previous release (v3.4) that broke use of `jdbc:` URLs.
- Fix regression bug in previous release (v3.4) affecting `text`, `ascii` and `ipac` output formats, that could cause cached table output (e.g. from filters `cache` or `random`) to contain many repeated copies of the same row.
- Upgrade cds-healpix-java library to v0.29.3; this avoids occasional sky crossmatch failures (termination with error) introduced at STILTS v3.2-1.
- Fix bug in `addcol` filter that could evaluate the row index (`$index`/`$0`) incorrectly during parallel processing.
- Fix `plot2sky` bug that got `clon`/`clat` positioning wrong for some projections such as `CAR0`.
- Fix issue with cumulative histograms; bars beyond the last sample are now displayed with total value not zero.
- Fix bugs that meant writing long (>2Gb) fits-var files could output incorrect/corrupted FITS.
- Fix long-standing file caching bug; mostly seemed to affect large (>2Gb) streams.
- Fix documentation bug in cone; output parameters `out`, `ofmt`, `ocmd` and `omode` are now listed.
- Fix optional argument processing bug in JyStilts (optional named arguments like `start` in `tloop` were being ignored).
- Fix tcube bug that caused problems with some input filter (`icmd`) processing.
- Fix long-standing logic error in ASCII/CSV input handler that could misidentify column types and cause read failures.
- Fix failure when trying to plot with `auxfunc=histogram` and explicitly set `auxmin`/`auxmax` bounds outside of data range.
- FITS TZERO headers are now written correctly with numeric values rather than string values.

Version 3.4-2 (15 October 2021)

Taplint and Votlint changes:

- New stage `UUC` added to taplint. This checks units (against the VOUnits standard) and UCDs (against the UCD1+ standard and legacy UCD1 list).
- New stage `EPN` added to taplint. This checks EPN-TAP (`*.epn_core`) tables against

some version of EPN-TAP 2.0 (currently PR-EPNTAP-2.0-20211012). Since EPN-TAP is still in draft at time of writing, this stage is not currently run by default.
- Modify taplint `stages` parameter usage; you can now add and remove stages individually using `+XXX`/`-XXX` syntax.
- Taplint now does a better job of categorising VOTable parse errors, so multiple repeats of similar messages can be avoided. Ad-hoc VOTable error codes have changed from "`VOxx`" to "`Yxxx`".
- Votlint, hence also taplint, now checks values in FIELD/PARAMs marked with `xtype` attributes for conformance to rules from DALI 1.1 sec 3.3.
- Taplint `MDQ` stage now reports mismatches between declared and result Units, UCDs, Xtypes, and Utypes as well as datatypes.
- Taplint and datalinklint now identify themselves as validators using the `User-Agent` token "`(IVOA-test)`" rather than "`(IVOA-validate)`" as before; this follows the published content of the SoftID-1.0 IVOA Note.
- Taplint now examines the service `Server` header and tries to check it for VO component identification as recommended by `SoftID`.
- Votlint now has a `maxrepeat` parameter like `taplint` (previously the repeat count was hard-wired to 4).
- Votlint now does a better job of identifying similar errors, so output suppressing repeated messages can be more compact. Suppression of repeated messages is reported.
- Votlint now optionally checks the content of `unit` and `ucd` attributes, checking against the VOUnits and UCD1+/UCD1 standards respectively.
- Distinguish `votlint` reports for recognised and unrecognised (e.g. VODML) elements in foreign XML namespaces.
- Add timestamp line at start of `taplint` output.

Other new functionality:

- A new parameter `thick` has been added to several plot layer types to enable lines thicker than a single pixel when drawing error bars, arrows, outlines, pair links etc: `xyvector`, `xyerror`, `sizexy`, `xyellipse`, `xycorr`, `link2`, `poly4`, `polygon`, `area`, `yerrors`, `xyerrors`, `skyvector`, `skyellipse`, `skycorr`, `xyzvector`, `xyzerror`, `yerror`.
- New functions in class VO to check syntax of UCD and VOUnit strings.
- New `areatype` value `UNIQ` for plotting single HEALPix tiles e.g. from MOC files in area (and similar) layer type.
- New `checksum` table output mode.
- Add Julian Day manipulation functions `jdToMjd` and `mjdToJd` to class Times.

Minor enhancements and behaviour changes:

- The `fast` parameter has been withdrawn from polygon-plotting plot types `area`, `polygon`, `poly4`; the shape-filling algorithm has been improved, and it's no longer necessary to choose between speed and accuracy.
- Replace the `julianToUnixSec` function in class Times, which didn't do what it said it did, with `jdToUnixSec`.
- The `mjdToIso` and `mjdToDate` functions in class Times now prepend the string "`(BCE)`" to dates before the Common Era.
- Guess meaning for some non-standard COORDSYS values in HEALPix-FITS files, e.g. allow "GALACTIC" instead of "G".
- Revert to sequential processing in some cases for HEALPix plot layer to reduce resource usage.
- Columns that are all blank in ASCII-like tables (CSV, ASCII, TST) are now interpreted as type String not boolean.
- Files compressed using multi-stream bzip2 compression (e.g. `pbzip2` output) are

now supported alongside single-stream bzip2.

Bugfixes:

- Fix serious threading bug that could return nonsense values from fixed-length string fields during parallel processing, for instance statistics calculation, of large cached or randomised tables.
- Upgrade of cds-healpix-java library to v0.30.2; this fixes some HEALPix cell plotting bugs.
- Fix aux ranging bug that meant `auxmax` without `auxmin` was not honoured in plot commands. This was a regression bug introduced at v3.1-5.

Version 3.4-3 (31 January 2022)

New functionality:

- The PDS4 (NASA's Planetary Data System v4) file format is now supported for input tables.
- New task tgridmap writes flexible N-dimensional density maps/histograms as tables.
- The `cols` parameter in tskymap is generalised to allow more flexible calculation of gridded quantities, specifically calculations using different aggregation methods for different quantities at the same time.
- New plot2plane layers statline and statmark, to plot quantities like the mean of array-valued input columns.
- New class Randoms contains pseudo-random number generation functions. Special token `$random` is now deprecated.
- New quantities `ArrayNGood`, `ArraySum`, `ArrayMean`, `ArrayStDev` added to `stats` filter, allowing per-element statistics calculations for fixed-length numeric array data.
- New taplint parameter `tables` to restrict which tables are tested.
- Provide quantile options for the `combiner` parameters of some commands.

Minor enhancements and behaviour changes:

- Add new colour map Painbow.
- Improve identification of TIME_TT2000 columns as time values in certain CDF files.
- Improved autoranging for gaussian plots.
- Minor improvements to aggregated column output metadata.
- JyStilts is now intended to be run using Jython 2.7.2, which means it works under Java 17, unlike Jython 2.5.3 which was the previous target version. The JyStilts classes haven't changed, but the distributed `jystilts.jar` file now comes with Jython 2.7.2.
- In the starjava distribution, the `jystilts` script now uses an internal copy of Jython 2.7.2, it no longer requires an external Jython installation.
- Taplint UUC stage now reports UCD word list versions.
- Adjustments to taplint EPN-TAP validation: track draft standard changes, fix bugs, add UCD check for measurement_type column content. This version currently corresponds to PR-EPNTAP-2.0-20211022.
- Make a couple of adjustments to SVG graphics output: output is now to a bare `svg` element (no XML or DOCTYPE declaration), and a `viewBox` attribute is included which may improve scaling behaviour in some contexts.
- The monolithic stilts.jar file and stilts_jars.zip archive now contain the `stilts.version` file at the top level, rather than just buried in `uk/ac/starlink/ttools/`.
- The `seqcol` column in output from the `tcat` and `tcatn` commands is now of type (32-bit) integer not (16-bit) short.

- Permit leading 's' in ASCII-MOC area specifications, following MOC 2.0.

Bug fixes:

- Fix bugs in parsing `MOC-ASCII` strings in area plot; trailing depth specifier was interpreted as cell index and some cells near end of MOC were omitted/misshapen.
- Fix failure when attempting to read unsigned 32-bit integer values from parquet files.
- Bugfix update of JCDF to v1.2-4.

Version 3.4-4 (6 April 2022)

Performance

- Substantial I/O performance improvements, mainly for FITS and VOTable formats e.g.: writing to FITS 2x, reading FITS from a stream 2x, reading VOTable with inline BINARY/BINARY2 4x, writing VOTable with inline BINARY/BINARY2 2x, writing VOTable with TABLEDATA 1.5x.
- New parameter `runner` in matching commands tmatch2 etc which can be set to `parallel` for multi-threaded execution. This should speed up large matches on multi-core machines. This is believed to work correctly, but since it is less well tested than the legacy sequential mode, at this release sequential processing is still the default.
- Other matching performance improvements.
- FITS I/O is now all done internally, there is no longer a dependency on the nom.tam.fits package.

New functionality

- New command `xsdvalidate` added for validating against XML schemas.
- New class Bits with bit manipulation functions `bitCount`, `hasBit`, `toBinary`, `fromBinary`.
- Add configuration options `compact` and `encoding` to VOTable output handler. By default thin (<=4 column) TABLEDATA VOTables are now written in "compact" mode, using reduced whitespace.
- FITS BINTABLE headers used as table parameters now support FITS 4.0 long-string syntax (CONTINUE records).
- FITS header values of the form "(a,b,c,...)" are now interpreted where possible as numeric arrays; this works for long-string values (CONTINUE records) as well.

Bug Fixes and workarounds

- Upgrade JEL to v2.1.3-pre1. This fixes a bug that caused evaluation failure when comparing a String against null.
- Taplint now reports absent TAP service with an error code rather than failing with an exception.
- Upgrade Unity library to 1.1 pre-release and improve VOUnits validation reporting (new status "GUESSED_UNIT").
- Fix taplint bug: EPN validation failed for `spatial_frame_type=none`.
- ECSV format now preserves table name.
- FITS BINTABLE reader now copes with (illegal?) embedded spaces in TDIMn headers.
- Adjust MRT null handling; "-" in a single-character field no longer interpreted as null.

Version 3.4-5 (10 June 2022)

New Functionality

- New plot2plane layer arrayquantile, that can plot e.g. medians of multiple per-row spectra with arbitrary wavelength coordinates.
- X/Y array-valued plot layer types lines, marks, yerrors, xyerrors, statline and statmark will now accept a missing `xs` or `ys` array coordinate, and assume a suitable linear sequence.
- The `<colid-list>` syntax used by `keepcols`, `delcols` etc now allows column ranges to be specified with the syntax `<first-colid>-<last-colid>`.
- The `assert` filter now takes optional `<msg-expr>` argument.
- Add offset options `xoff` and `yoff` for `label` plot layer type.
- Minor `taplint` EPN stage updates corresponding to latest EPN-TAP draft; now matches PR-EPNTAP-20220420.

Performance

- Sky crossmatching performance improvements.
- Reduce number of file mapping calls by FITS readers.

Bug Fixes

- Fix FITS parsing issue that could result in StackOverflowError for long array-valued headers.
- Fix bugs in statline/statmark layers that incorrectly treated blank array values and negative array values on logarithmic axes.
- Fix bug in multi-threaded read of string columns from colfits files.
- Fix NullPointerException failure in taplint UUC stage or votlint for unknown units recognised only in non-VOUnit syntaxes (e.g. "Crab").
- Fix legend positioning bug for `label` layer type.
- Aitoff sky projections are now more correctly documented as Hammer-Aitoff (though not renamed in the UI).

Version 3.4-6 (8 July 2022)

Enhancements

- New command `arrayjoin` added.
- Modify functions `add`, `subtract`, `multiply`, `divide` in class Arrays; these now all take either two array arguments or an array and a scalar in either order.
- New function `dotProduct` in class Arrays.
- Provide `value*` functions for column references to strangely-named columns.
- Minor `taplint` EPN stage update; now corresponds to PR-EPNTAP-20220705.

Bug Fix

- Fix FITS output so it doesn't fail when attempting to write metadata with non-ASCII Unicode characters.