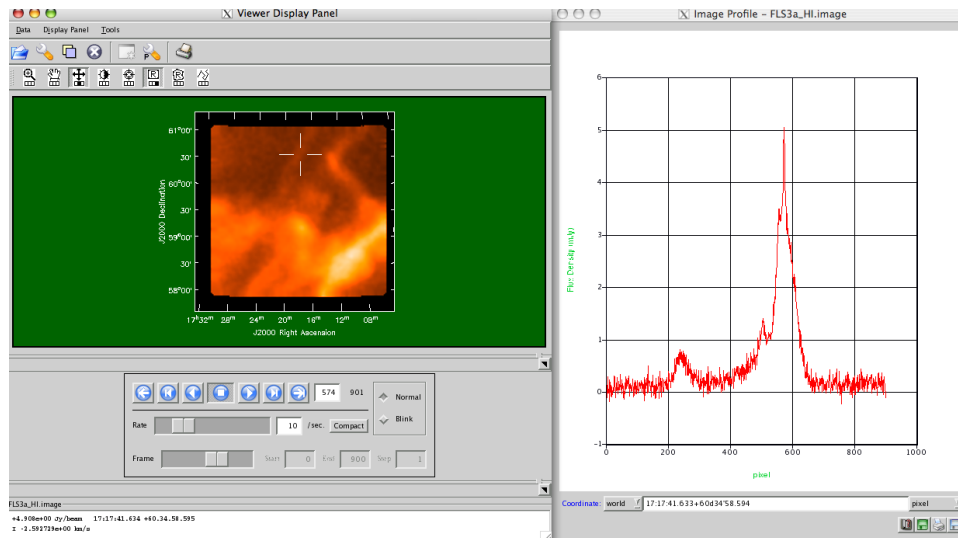
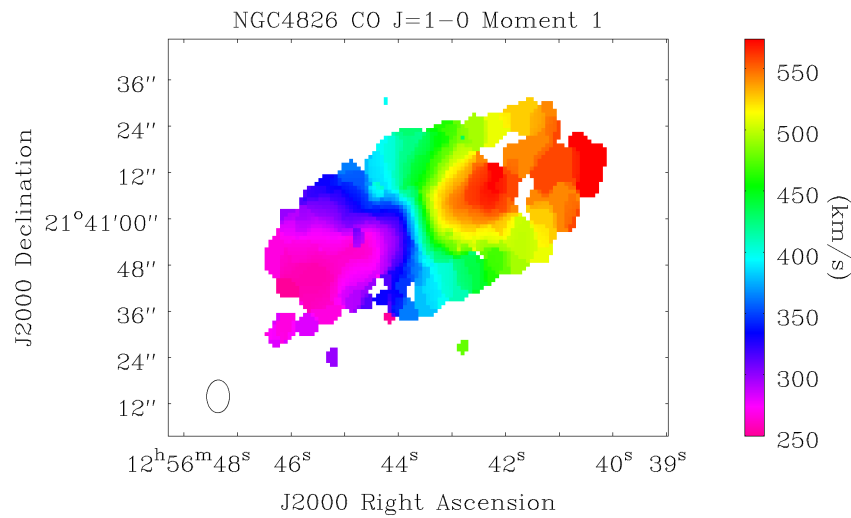


CASA User Reference & Cookbook

Release 3.0.2



Version: June 8, 2010

CASA Synthesis & Single Dish Reduction Cookbook

Editors: Steven T. Myers & Jürgen Ott – Project Scientist
Nick Elias – CASA Project Manager

Do you dare to enter CASA Stadium and join battle with the Ironic Chefs?
Let us see whose cuisine reigns supreme . . .

CASA SYNTHESIS & SINGLE DISH REDUCTION COOKBOOK,
Version June 8, 2010,
©2010 National Radio Astronomy Observatory

The National Radio Astronomy Observatory is a facility of the National Science Foundation
operated under cooperative agreement by Associated Universities, Inc.

Permission is granted to make and distribute verbatim copies of this manual provided the copyright notice and this permission notice are preserved on all copies.

Permission is granted to copy and distribute modified versions of this manual under the conditions for verbatim copying, provided that the entire resulting derived work is distributed under the terms of a permission notice identical to this one.

Permission is granted to copy and distribute translations of this manual into another language, under the above conditions for modified versions, except that this permission notice may be stated in a translation approved by AUI.

For help, comments, or corrections, please visit <http://help.nrao.edu>.

Contents

1	Introduction	21
1.1	About This Release	23
1.1.1	What's New in Release 3.0.1	23
1.2	CASA Basics — Information for First-Time Users	27
1.2.1	Before Starting CASA	28
1.2.1.1	Environment Variables	29
1.2.1.2	Where is CASA?	29
1.2.2	Starting CASA	30
1.2.3	Ending CASA	30
1.2.4	What happens if something goes wrong?	30
1.2.5	Aborting CASA execution	31
1.2.6	What happens if CASA crashes?	31
1.2.7	Python Basics for CASA	32
1.2.7.1	Variables	32
1.2.7.2	Lists and Ranges	32
1.2.7.3	Indexes	33
1.2.7.4	Indentation	33
1.2.7.5	System shell access	33
1.2.7.6	Executing Python scripts	34
1.2.8	Getting Help in CASA	34
1.2.8.1	TAB key	34
1.2.8.2	help <taskname>	35
1.2.8.3	help and PAGER	37
1.2.8.4	help par.<parameter>	37
1.2.8.5	Python help	37
1.3	Tasks and Tools in CASA	38
1.3.1	What Tasks are Available?	39
1.3.2	Running Tasks and Tools	43
1.3.2.1	Aborting Synchronous Tasks	44
1.3.3	Getting Return Values	45
1.3.4	Running Tasks Asynchronously	46
1.3.4.1	Monitoring Asynchronous Tasks	46
1.3.4.2	Aborting Asynchronous Tasks	48
1.3.5	Setting Parameters and Invoking Tasks	48

1.3.5.1	The scope of parameters in CASA	49
1.3.5.2	The <code>default</code> Command	50
1.3.5.3	The <code>go</code> Command	51
1.3.5.4	The <code>inp</code> Command	52
1.3.5.5	The <code>saveinputs</code> Command	55
1.3.5.6	The <code>tget</code> Command	56
1.3.5.7	The <code>tput</code> Command	57
1.3.5.8	The <code>.last</code> file	57
1.3.6	Tools in CASA	58
1.4	Getting the most out of CASA	59
1.4.1	Your command line history	59
1.4.2	Logging your session	59
1.4.2.1	Startup options for the <code>logger</code>	62
1.4.2.2	Setting priority levels in the <code>logger</code>	63
1.4.3	Where are my data in CASA?	64
1.4.3.1	How do I get rid of my data in CASA?	64
1.4.4	What's in my data?	65
1.4.5	Data Selection in CASA	65
1.5	From Loading Data to Images	66
1.5.1	Loading Data into CASA	67
1.5.1.1	VLA: Filling data from VLA archive format	67
1.5.1.2	Filling data from UVFITS format	67
1.5.1.3	Loading FITS images	67
1.5.1.4	Concatenation of multiple MS	68
1.5.2	Data Examination, Editing, and Flagging	68
1.5.2.1	Interactive X-Y Plotting and Flagging	68
1.5.2.2	Flag the Data Non-interactively	69
1.5.2.3	Viewing and Flagging the MS	69
1.5.3	Calibration	69
1.5.3.1	Prior Calibration	70
1.5.3.2	Bandpass Calibration	70
1.5.3.3	Gain Calibration	70
1.5.3.4	Polarization Calibration	71
1.5.3.5	Examining Calibration Solutions	71
1.5.3.6	Bootstrapping Flux Calibration	71
1.5.3.7	Correcting the Data	71
1.5.3.8	Splitting the Data	71
1.5.3.9	UV Continuum subtraction	72
1.5.3.10	Transforming the Data to a new frame	72
1.5.4	Synthesis Imaging	72
1.5.4.1	Cleaning a single-field image or a mosaic	72
1.5.4.2	Feathering in a Single-Dish image	73
1.5.5	Self Calibration	73
1.5.6	Data and Image Analysis	73
1.5.6.1	What's in an image?	74

1.5.6.2	Image statistics	74
1.5.6.3	Image values	74
1.5.6.4	Moments of an image cube	74
1.5.6.5	Image math	74
1.5.6.6	Regridding an Image	75
1.5.6.7	Displaying Images	75
1.5.7	Getting data and images out of CASA	75
2	Visibility Data Import, Export, and Selection	76
2.1	CASA Measurement Sets	77
2.1.1	Under the Hood: Structure of the Measurement Set	77
2.2	Data Import and Export	80
2.2.1	ALMA & EVLA: Filling Science Data Model (ASDM) data	80
2.2.1.1	Filling SDM v0 data using <code>importoldasdm</code>	81
2.2.2	VLA: Filling data from archive format (<code>importvla</code>)	82
2.2.2.1	Parameter <code>applysys</code>	83
2.2.2.2	Parameter <code>bandname</code>	83
2.2.2.3	Parameter <code>frequencytol</code>	84
2.2.2.4	Parameter <code>project</code>	84
2.2.2.5	Parameters <code>starttime</code> and <code>stoptime</code>	84
2.2.2.6	Parameter <code>autocorr</code>	84
2.2.2.7	Parameter <code>antnamescheme</code>	85
2.2.2.8	Parameter <code>evlabands</code>	85
2.2.3	UVFITS Import and Export	85
2.2.3.1	Import using <code>importuvfits</code>	85
2.2.3.2	Export using <code>exportuvfits</code>	86
2.2.4	Handling Measurement Set metadata and data	87
2.2.5	Summarizing your MS (<code>listobs</code>)	87
2.2.6	Listing MS data (<code>listvis</code>)	89
2.2.7	Listing and manipulating MS metadata (<code>vishead</code>)	90
2.2.8	MS statistics (<code>visstat</code>)	92
2.2.9	Concatenating multiple datasets (<code>concat</code>)	93
2.3	Data Selection	94
2.3.1	General selection syntax	95
2.3.1.1	String Matching	96
2.3.2	The <code>field</code> Parameter	97
2.3.3	The <code>spw</code> Parameter	97
2.3.3.1	Channel selection in the <code>spw</code> parameter	98
2.3.4	The <code>selectdata</code> Parameters	99
2.3.4.1	The <code>antenna</code> Parameter	100
2.3.4.2	The <code>scan</code> Parameter	101
2.3.4.3	The <code>timerange</code> Parameter	102
2.3.4.4	The <code>uvrange</code> Parameter	103
2.3.4.5	The <code>msselect</code> Parameter	104

3	Data Examination and Editing	105
3.1	Plotting and Flagging Visibility Data in CASA	105
3.2	Managing flag versions with <code>flagmanager</code>	106
3.3	Flagging auto-correlations with <code>flagautocorr</code>	107
3.4	X-Y Plotting and Editing of the Data	108
3.4.1	MS Plotting and Editing using <code>plotms</code>	108
3.4.1.1	Loading and Selecting Data	108
3.4.1.2	Plot Axes	110
3.4.1.3	Tools	113
3.4.1.4	Interactive Flagging in <code>plotms</code>	114
3.4.1.5	Averaging Data	115
3.4.1.6	Plot Symbols	116
3.4.1.7	Summarizing Data	117
3.4.1.8	Defining Frequency and Velocity	117
3.4.1.9	Shifting the Phase Center	118
3.4.1.10	Plot Ranges	119
3.4.1.11	Plot Labels	119
3.4.1.12	Grid Lines	119
3.4.1.13	Legend	119
3.4.1.14	New Plot Windows and Plot Selection	120
3.4.1.15	The Cache	121
3.4.1.16	Iteration	121
3.4.1.17	Saving your plot	121
3.4.1.18	Exiting <code>plotms</code>	122
3.4.2	Plotting and Editing using <code>plotxy</code>	122
3.4.2.1	GUI Plot Control	125
3.4.2.2	The <code>selectplot</code> Parameters	126
3.4.2.3	The <code>iteration</code> parameter	127
3.4.2.4	The <code>overplot</code> parameter	128
3.4.2.5	The <code>plotrange</code> parameter	129
3.4.2.6	The <code>plotsymbol</code> parameter	129
3.4.2.7	The <code>showflags</code> parameter	130
3.4.2.8	The <code>subplot</code> parameter	130
3.4.2.9	Averaging in <code>plotxy</code>	131
3.4.2.10	Interactive Flagging in <code>plotxy</code>	131
3.4.2.11	Flag extension in <code>plotxy</code>	133
3.4.2.12	Setting rest frequencies in <code>plotxy</code>	134
3.4.2.13	Printing from <code>plotxy</code>	136
3.4.2.14	Exiting <code>plotxy</code>	136
3.4.2.15	Example session using <code>plotxy</code>	137
3.4.3	Plotting antenna positions using <code>plotants</code>	139
3.5	Non-Interactive Flagging using <code>flagdata</code>	140
3.5.1	Flag Antenna/Channels	141
3.5.1.1	Manual flagging and clipping in <code>flagdata</code>	142
3.5.1.2	Flagging the beginning of scans	143

3.5.1.3	Flagging shadowed data with mode 'shadow'	143
3.5.1.4	Autoflagging using modes 'autoflag' and 'rfi'	144
3.6	Browse the Data	144
4	Synthesis Calibration	148
4.1	Calibration Tasks	148
4.2	The Calibration Process — Outline and Philosophy	150
4.2.1	The Philosophy of Calibration in CASA	152
4.2.2	Keeping Track of Calibration Tables	152
4.2.3	The Calibration of VLA data in CASA	153
4.3	Preparing for Calibration	154
4.3.1	System Temperature Correction	155
4.3.2	Antenna Gain-Elevation Curve Calibration	155
4.3.3	Atmospheric Optical Depth Correction	156
4.3.3.1	Determining opacity corrections for VLA data	157
4.3.4	Setting the Flux Density Scale using (<code>setjy</code>)	157
4.3.4.1	Using Calibration Models for Resolved Sources	159
4.3.5	Correction for delay and antenna position offsets using <code>genca1</code>	161
4.3.6	Other <i>a priori</i> Calibrations and Corrections	164
4.4	Solving for Calibration — Bandpass, Gain, Polarization	164
4.4.1	Common Calibration Solver Parameters	164
4.4.1.1	Parameters for Specification: <code>vis</code> and <code>caltable</code>	165
4.4.1.2	Selection: <code>field</code> , <code>spw</code> , and <code>selectdata</code>	165
4.4.1.3	Prior Calibration and Correction: <code>parang</code> , <code>gaincurve</code> and <code>opacity</code>	166
4.4.1.4	Previous Calibration: <code>gaintable</code> , <code>gainfield</code> , <code>interp</code> and <code>spwmap</code>	166
4.4.1.5	Solving: <code>solint</code> , <code>combine</code> , <code>preavg</code> , <code>refant</code> , <code>minblperant</code> , <code>minsnr</code>	168
4.4.1.6	Action: <code>append</code> and <code>solnorm</code>	169
4.4.2	Spectral Bandpass Calibration (<code>bandpass</code>)	169
4.4.2.1	Bandpass Normalization	170
4.4.2.2	B solutions	171
4.4.2.3	BPOLY solutions	172
4.4.3	Complex Gain Calibration (<code>gaincal</code>)	173
4.4.3.1	Polarization-dependent Gain (G)	174
4.4.3.2	Polarization-independent Gain (T)	175
4.4.3.3	GSPLINE solutions	176
4.4.4	Establishing the Flux Density Scale (<code>fluxscale</code>)	177
4.4.4.1	Using Resolved Calibrators	179
4.4.5	Instrumental Polarization Calibration (D,X)	180
4.4.5.1	Heuristics and Strategies for Polarization Calibration	181
4.4.5.2	A Polarization Calibration Example	182
4.4.6	Baseline-based Calibration (<code>blcal</code>)	183
4.5	Plotting and Manipulating Calibration Tables	184
4.5.1	Plotting Calibration Solutions (<code>plotcal</code>)	184
4.5.1.1	Examples for <code>plotcal</code>	186
4.5.2	Listing calibration solutions with (<code>listcal</code>)	188

4.5.3	Calibration table statistics (<code>calstat</code>)	190
4.5.4	Calibration Smoothing (<code>smoothcal</code>)	191
4.5.5	Calibration Interpolation and Accumulation (<code>accum</code>)	192
4.5.5.1	Interpolation using (<code>accum</code>)	194
4.5.5.2	Incremental Calibration using (<code>accum</code>)	195
4.6	Application of Calibration to the Data	199
4.6.1	Application of Calibration (<code>applycal</code>)	199
4.6.2	Examine the Calibrated Data	201
4.6.3	Resetting the Applied Calibration using (<code>clearcal</code>)	201
4.7	Other Calibration and UV-Plane Analysis Options	202
4.7.1	Splitting out Calibrated uv data (<code>split</code>)	202
4.7.1.1	Averaging in <code>split</code>	203
4.7.2	Hanning smoothing of uv data (<code>hanningsmooth</code>)	204
4.7.3	Model subtraction from uv data (<code>uvsub</code>)	204
4.7.4	UV-Plane Continuum Subtraction (<code>uvcontsub</code>)	205
4.7.5	Spectral regridding of the MS (<code>cvel</code>)	207
4.7.6	UV-Plane Model Fitting (<code>uvmodelfit</code>)	209
4.8	Examples of Calibration	211
5	Synthesis Imaging	213
5.1	Imaging Tasks Overview	213
5.2	Common Imaging Task Parameters	214
5.2.1	Parameter <code>cell</code>	214
5.2.2	Parameter <code>field</code>	215
5.2.3	Parameter <code>imagename</code>	215
5.2.4	Parameter <code>imsize</code>	215
5.2.5	Parameter <code>mode</code>	215
5.2.5.1	Mode <code>mfs</code>	216
5.2.5.2	Mode <code>channel</code>	216
5.2.5.3	Mode <code>frequency</code>	217
5.2.5.4	Mode <code>velocity</code>	218
5.2.5.5	Sub-parameter <code>interpolation</code>	219
5.2.6	Parameter <code>phasecenter</code>	219
5.2.7	Parameter <code>restfreq</code>	219
5.2.8	Parameter <code>spw</code>	220
5.2.9	Parameter <code>stokes</code>	220
5.2.10	Parameter <code>uvtaper</code>	221
5.2.11	Parameter <code>weighting</code>	221
5.2.11.1	'natural' weighting	222
5.2.11.2	'uniform' weighting	222
5.2.11.3	'superuniform' weighting	222
5.2.11.4	'radial' weighting	223
5.2.11.5	'briggs' weighting	223
5.2.11.6	'briggsabs' weighting	223
5.2.12	Parameter <code>vis</code>	224

5.2.13	Primary beams in imaging	224
5.3	Deconvolution using CLEAN (<code>clean</code>)	225
5.3.1	Parameter <code>psfmode</code>	228
5.3.1.1	The <code>clark</code> algorithm	228
5.3.1.2	The <code>hogbom</code> algorithm	228
5.3.1.3	The <code>clarkstokes</code> algorithm	228
5.3.2	The <code>multiscale</code> parameter	229
5.3.3	Parameter <code>gain</code>	229
5.3.4	Parameter <code>imagermode</code>	230
5.3.4.1	Sub-parameter <code>cyclefactor</code>	231
5.3.4.2	Sub-parameter <code>cyclespeedup</code>	232
5.3.4.3	Sub-parameter <code>ftmachine</code>	232
5.3.4.4	Sub-parameter <code>mosweight</code>	233
5.3.4.5	Sub-parameter <code>scaletype</code>	233
5.3.4.6	The <code>threshold</code> revisited	233
5.3.5	Parameter <code>interactive</code>	234
5.3.6	Parameter <code>mask</code>	234
5.3.6.1	Setting clean boxes	235
5.3.6.2	Using clean box files	235
5.3.6.3	Using clean mask images	235
5.3.6.4	Using region files	235
5.3.7	Parameter <code>minpb</code>	236
5.3.8	Parameter <code>modelimage</code>	236
5.3.9	Parameter <code>niter</code>	236
5.3.10	Parameter <code>pbcor</code>	236
5.3.11	Parameter <code>restoringbeam</code>	237
5.3.12	Parameter <code>threshold</code>	237
5.3.13	Parameter <code>gridmode</code>	237
5.3.14	Interactive Cleaning — Example	238
5.3.15	Mosaic imaging	241
5.3.16	Heterogeneous imaging	244
5.3.17	Polarization imaging	244
5.3.18	Wide-field imaging and deconvolution in <code>clean</code>	244
5.3.18.1	Outlier fields	245
5.3.18.2	Setting up w-projection	246
5.3.18.3	Setting up faceting	246
5.3.18.4	Combination of w-projection and faceting	247
5.4	Combined Single Dish and Interferometric Imaging (<code>feather</code>)	247
5.5	Making Deconvolution Masks or Box Regions	248
5.5.1	Making Deconvolution Regions from an Image (<code>boxit</code>)	248
5.5.2	Making Deconvolution Masks from Boxes (<code>makemask</code>)	249
5.6	Transforming an Image Model (<code>ft</code>)	251
5.7	Image-plane deconvolution (<code>deconvolve</code>)	252
5.8	Self-Calibration	252
5.9	Automatic Cleaning	253

5.10	Examples of Imaging	255
6	Image Analysis	256
6.1	Common Image Analysis Task Parameters	257
6.1.1	Region Selection (<code>box</code>)	257
6.1.2	Plane Selection (<code>chans</code> , <code>stokes</code>)	257
6.1.3	Lattice Expressions (<code>expr</code>)	258
6.1.4	Masks (<code>mask</code>)	259
6.1.5	Regions (<code>region</code>)	259
6.2	Image Header Manipulation (<code>imhead</code>)	260
6.2.1	Examples for <code>imhead</code>	261
6.3	Continuum Subtraction on an Image Cube (<code>imcontsub</code>)	263
6.3.1	Examples for <code>imcontsub</code>)	263
6.4	Image-plane Component Fitting (<code>imfit</code>)	264
6.4.1	Examples for <code>imfit</code>	265
6.5	Mathematical Operations on an Image (<code>immath</code>)	267
6.5.1	Examples for <code>immath</code>	268
6.5.1.1	Simple math	268
6.5.1.2	Polarization manipulation	270
6.5.1.3	Primary beam correction/uncorrection	271
6.5.1.4	Spectral analysis	272
6.5.2	Using masks in <code>immath</code>	272
6.6	Computing the Moments of an Image Cube (<code>immoments</code>)	274
6.6.1	Hints for using (<code>immoments</code>)	276
6.6.2	Examples using (<code>immoments</code>)	276
6.7	Computing image statistics (<code>imstat</code>)	277
6.7.1	Using the task return value	277
6.7.2	Examples for <code>imstat</code>)	280
6.8	Extracting data from an image (<code>imval</code>)	281
6.9	Regridding an Image (<code>imregrid</code>)	283
6.10	Image Convolution(<code>imsmooth</code>)	284
6.11	Image Import/Export to FITS	284
6.11.1	FITS Image Export (<code>exportfits</code>)	284
6.11.2	FITS Image Import (<code>importfits</code>)	285
6.12	Using the CASA Toolkit for Image Analysis	285
6.13	Examples of CASA Image Analysis	288
7	Visualization With The CASA Viewer	289
7.1	Starting the viewer	289
7.1.1	Running the CASA viewer outside <code>casapy</code>	291
7.2	The viewer GUI	292
7.2.1	The Viewer Display Panel	292
7.2.2	Saving and Restoring Display Panel State	296
7.2.3	Region Selection and Positioning	296
7.2.4	The Load Data Panel	297

7.2.4.1	Registered vs. Open Datasets	299
7.3	Viewing Images	299
7.3.1	Viewing a raster map	299
7.3.1.1	Raster Image — Display Axes	301
7.3.1.2	Raster Image — Basic Settings	301
7.3.1.3	Raster Image — Other Settings	304
7.3.2	Viewing a contour map	304
7.3.3	Overlay contours on a raster map	306
7.3.4	Spectral Profile Plotting	306
7.3.5	Managing and Saving Regions	307
7.3.6	Adjusting Canvas Parameters/Multi-panel displays	309
7.3.6.1	Setting up multi-panel displays	310
7.3.6.2	Background Color	310
7.4	Viewing Measurement Sets	311
7.4.1	Data Display Options Panel for Measurement Sets	311
7.4.1.1	MS Options — Basic Settings	312
7.4.1.2	MS Options— MS and Visibility Selections	313
7.4.1.3	MS Options — Display Axes	314
7.4.1.4	MS Options — Flagging Options	316
7.4.1.5	MS Options— Advanced	318
7.4.1.6	MS Options — Apply Button	319
7.5	Printing from the Viewer	319
A	Appendix: Single Dish Data Processing	322
A.1	Guidelines for Use of ASAP and SDtasks in CASA	323
A.1.1	Environment Variables	323
A.1.2	Assignment	323
A.1.3	Lists	323
A.1.4	Dictionaries	324
A.1.5	Line Formatting	325
A.1.6	Logging	325
A.2	Single Dish Analysis Tasks	325
A.2.1	SDtask Summaries	328
A.2.1.1	<code>sdaverage</code>	328
A.2.1.2	<code>sdsMOOTH</code>	332
A.2.1.3	<code>sdbaseline</code>	333
A.2.1.4	<code>sdcal</code>	337
A.2.1.5	<code>sdcoadd</code>	341
A.2.1.6	<code>sdflag</code>	343
A.2.1.7	<code>sdFIT</code>	345
A.2.1.8	<code>sdIMAGING</code>	348
A.2.1.9	<code>sdLIST</code>	350
A.2.1.10	<code>sdMATH</code>	351
A.2.1.11	<code>sdPLOT</code>	354
A.2.1.12	<code>sdsAVE</code>	358

A.2.1.13	<code>sdscale</code>	360
A.2.1.14	<code>sdstat</code>	361
A.2.1.15	<code>sctpimaging</code>	364
A.2.1.16	<code>sdimprocess</code>	366
A.2.1.17	<code>sdsim</code>	367
A.2.2	Single Dish Analysis Use Cases With SDTasks	370
A.2.2.1	GBT Position Switched Data Analysis	370
A.2.2.2	Imaging of Total Power Raster Scans	383
A.3	Using The ASAP Toolkit Within CASA	384
A.3.1	Environment Variables	386
A.3.2	Import	387
A.3.2.1	General descriptions	387
A.3.2.2	Handling ALMA data	388
A.3.3	Scantable Manipulation	390
A.3.3.1	Data Selection	390
A.3.3.2	State Information	390
A.3.3.3	Masks	392
A.3.3.4	Scantable Management	392
A.3.3.5	Scantable Mathematics	393
A.3.3.6	Scantable Save and Export	393
A.3.4	Calibration	393
A.3.4.1	Tsys scaling	394
A.3.4.2	Flux and Temperature Unit Conversion	394
A.3.4.3	Gain-Elevation and Atmospheric Optical Depth Corrections	394
A.3.4.4	Calibration of GBT data	395
A.3.4.5	Comprehensive calibration function	396
A.3.5	Averaging	396
A.3.6	Spectral Smoothing	397
A.3.7	Baseline Fitting	398
A.3.8	Line Fitting	399
A.3.9	Plotting	399
A.3.9.1	ASAP plotter	399
A.3.9.2	Line Catalog	400
A.3.10	Setting/Getting Rest Frequencies	401
A.3.11	Single Dish Spectral Analysis Use Case With ASAP Toolkit	401
A.4	Single Dish Imaging	405
A.4.1	Single Dish Imaging Use Case With ASAP Toolkit	405
A.5	Known Issues, Problems, Deficiencies and Features	408
B	Appendix: Simulation	410
B.1	Simulating ALMA with <code>simdata</code>	410
C	Appendix: Obtaining and Installing CASA	413
C.1	Installation Script	413
C.2	Startup	413

D	Appendix: Python and CASA	414
D.1	Automatic parentheses	414
D.2	Indentation	415
D.3	Lists and Ranges	415
D.4	Dictionaries	416
D.4.1	Saving and Reading Dictionaries	416
D.5	Control Flow: Conditionals, Loops, and Exceptions	418
D.5.1	Conditionals	418
D.5.2	Loops	420
D.6	System shell access	421
D.6.1	Using the <code>os.system</code> methods	421
D.6.2	Directory Navigation	423
D.6.3	Shell Command and Capture	423
D.7	Logging	425
D.8	History and Searching	425
D.9	Macros	427
D.10	On-line editing	428
D.11	Executing Python scripts	428
D.12	How do I exit from CASA?	429
E	Appendix: The Measurement Equation and Calibration	430
E.1	The HBS Measurement Equation	430
E.2	General Calibrator Mechanics	434
F	Appendix: Annotated Example Scripts	435
F.1	NGC 5921 — VLA red-shifted HI emission	435
F.2	Jupiter — VLA continuum polarization	460
F.3	BIMA Mosaic Spectral Imaging	499
G	Appendix: CASA Dictionaries	520
G.1	AIPS – CASA dictionary	520
G.2	MIRIAD – CASA dictionary	520
G.3	CLIC – CASA dictionary	520
H	Appendix: Writing Tasks in CASA	523
H.1	The XML file	524
H.2	The <code>task_yourtask.py</code> file	527
H.3	Example: The <code>clean</code> task	527
H.3.1	File <code>clean.xml</code>	527
H.3.2	File <code>task_clean.py</code>	545

List of Tables

2.1	Common MS Columns	79
2.2	Commonly accessed MAIN Table columns	80
4.1	Recognized Flux Density Calibrators.	158
G.1	MIRIAD – CASA dictionary	521
G.2	CLIC–CASA dictionary	522

List of Figures

1.1	Screen shot of the default CASA inputs for task <code>clean</code>	53
1.2	The <code>clean</code> inputs after setting values away from their defaults (blue text). Note that some of the boldface ones have opened up new dependent sub-parameters (indented and green).	54
1.3	The <code>clean</code> inputs where one parameter has been set to an invalid value. This is drawn in red to draw attention to the problem. This hapless user probably confused the 'hogbom' <code>clean</code> algorithm with Harry Potter.	55
1.4	The CASA Logger GUI window under Linux. Note that under MacOSX a stripped down logger will instead appear as a Console.	59
1.5	Using the Search facility in the <code>casalogger</code> . Here we have specified the string 'plotted' and it has highlighted all instances in green.	60
1.6	Using the <code>casalogger</code> Filter facility. The log output can be sorted by Priority, Time, Origin, and Message. In this example we are filtering by Origin using 'clean', and it now shows all the log output from the <code>clean</code> task.	61
1.7	CASA Logger - Insert facility: The log output can be augmented by adding notes or comments during the reduction. The file should then be saved to disk to retain these changes.	62
1.8	Flow chart of the data processing operations that a general user will carry out in an end-to-end CASA reduction session.	66
2.1	The contents of a Measurement Set. These tables compose a Measurement Set named <code>ngc5921.demo.ms</code> on disk. This display is obtained by using the File:Open menu in <code>browsetable</code> and left double-clicking on the <code>ngc5921.demo.ms</code> directory.	78
3.1	A freshly-started <code>plotms</code> GUI window. Note that the Plots > Data tab is selected, which is discussed in § 3.4.1.1, 3.4.1.5, and 3.4.1.7.	109
3.2	The Plots > Axes tab in the <code>plotms</code> GUI window, used to make a plot of Amp versus Channel	110
3.3	Plot of amplitude versus time, before (left) and after (right) flagging two marked regions. To unflag these regions, mark the two same regions and click the Unflag button.	115
3.4	Multiple "single" plots displayed in a <code>plotms</code> window. The Plot Selection menu is circled in red (See § 3.4.1.14 for more detail).	120

3.5	The <code>plotxy</code> plotter, showing the Jupiter data versus uv-distance. You can see bad data in this plot. The bottom set of buttons on the lower left are: 1,2,3) Home, Back, and Forward . Click to navigate between previously defined views (akin to web navigation). 4) Pan . Click and drag to pan to a new position. 5) Zoom . Click to define a rectangular region for zooming. 6) Subplot Configuration . Click to configure the parameters of the subplot and spaces for the figures. 7) Save . Click to launch a file save dialog box. The upper set of buttons in the lower left are: 1) Mark Region . Press this to begin marking regions (rather than zooming or panning). 2,3,4) Flag, Unflag, Locate . Click on these to flag, unflag, or list the data within the marked regions. 5) Next . Click to move to the next in a series of iterated plots. Finally, the cursor readout is on the bottom right.	123
3.6	The <code>plotxy</code> iteration plot. The first set of plots from the example in § 3.4.2.3 with <code>iteration='antenna'</code> . Each time you press the Next button, you get the next series of plots.	128
3.7	Multi-panel display of visibility versus channel (top), antenna array configuration (bottom left) and the resulting uv coverage (bottom right). The commands to make these three panels respectively are: 1) <code>plotxy('ngc5921.ms', xaxis='channel', datacolumn='data', field='0', subplot=211, plotcolor='', plotsymbol='go')</code> 2) <code>plotxy('ngc5921.ms', xaxis='x', field='0', subplot=223, plotsymbol='r.')</code> 3) <code>plotxy('ngc5921.ms', xaxis='u', yaxis='v', field='0', subplot=224, plotsymbol='b.', figure=1)</code>	132
3.8	Plot of amplitude versus uv distance, before (left) and after (right) flagging two marked regions. The call was: <code>plotxy(vis='ngc5921.ms', xaxis='uvdist', field='1445*')</code>	133
3.9	<code>flagdata</code> : Example showing before and after displays using a selection of one antenna and a range of channels. Note that each invocation of the <code>flagdata</code> task represents a cumulative selection, i.e., running <code>antenna='0'</code> will flag all data with antenna 0, while <code>antenna='0', spw='0:10 15'</code> will flag only those channels on antenna 0.	142
3.10	<code>flagdata</code> : Flagging example using the clip facility.	143
3.11	<code>browsetable</code> : The browser displays the main table within a frame. You can scroll through the data (x=columns of the MAIN table, and y=the rows) or select a specific page or row as desired. By default, 1000 rows of the table are loaded at a time, but you can step through the MS in batches.	145
3.12	<code>browsetable</code> : You can use the tab for Table Keywords to look at other tables within an MS. You can then double-click on a table to view its contents.	146
3.13	<code>browsetable</code> : Viewing the SOURCE table of the MS.	147
4.1	Flow chart of synthesis calibration operations. Not shown are use of table manipulation and plotting tasks <code>accum</code> , <code>plotcal</code> , and <code>smoothcal</code> (see Figure 4.2).	151
4.2	Chart of the table flow during calibration. The parameter names for input or output of the tasks are shown on the connectors. Note that from the output solver through the accumulator only a single calibration type (e.g. 'B', 'G') can be smoothed, interpolated or accumulated at a time. The final set of cumulative calibration tables of all types are then input to <code>applycal</code> as shown in Figure 4.1.	153

4.3	Display of the amplitude (upper) and phase (lower) gain solutions for all antennas and polarizations in the <code>ngc5921</code> post-fluxscale table.	187
4.4	Display of the amplitude (upper), phase (middle), and signal-to-noise ratio (lower) of the <code>bandpass 'B'</code> solutions for <code>antenna='0'</code> and both polarizations for <code>ngc5921</code> . Note the falloff of the SNR at the band edges in the lower panel.	188
4.5	Display of the amplitude of the <code>bandpass 'B'</code> solutions. Iteration over antennas was turned on using <code>iteration='antenna'</code> . The first page is shown. The user would use the Next button to advance to the next set of antennas.	189
4.6	The <code>'amp'</code> of gain solutions for <code>NGC4826</code> before (top) and after (bottom) smoothing with a 7200 sec <code>smoothtime</code> and <code>smoothtype='mean'</code> . Note that the first solution is in a different <code>spw</code> and on a different source, and is not smoothed together with the subsequent solutions.	193
4.7	The <code>'phase'</code> of gain solutions for <code>NGC4826</code> before (top) and after (bottom) <code>'linear'</code> interpolation onto a 20 sec <code>accumtime</code> grid. The first scan was 3C273 in <code>spw='0'</code> while the calibrator scans on 1331+305 were in <code>spw='1'</code> . The use of <code>spwmap</code> was necessary to transfer the interpolation correctly onto the <code>NGC4826</code> scans.	196
4.8	The final <code>'amp'</code> (top) and <code>'phase'</code> (bottom) of the self-calibration gain solutions for Jupiter. An initial phase calibration on 10s <code>solint</code> was followed by an incremental gain solution on each scan. These were accumulated into the cumulative solution shown here.	198
4.9	The final <code>'amp'</code> versus <code>'uvdist'</code> plot of the self-calibrated Jupiter data, as shown in <code>plotxy</code> . The <code>'RR LL'</code> correlations are selected. No outliers that need flagging are seen.	202
4.10	Use of <code>plotxy</code> to display corrected data (red and blue points) and uv model fit data (green circles).	212
5.1	Close-up of the top of the interactive <code>clean</code> window. Note the boxes at the left (where the <code>iterations</code> , <code>cycles</code> , and <code>threshold</code> can be changed), the buttons that control add/erase, the application of mask to channels, and whether to stop, complete, or continue cleaning, and the row of Mouse-button tool assignment icons.	230
5.2	Screen-shots of the interactive <code>clean</code> window during deconvolution of the VLA 6m Jupiter dataset. We start from the calibrated data, but before any self-calibration. In the initial stage (left), the window pops up and you can see it dominated by a bright source in the center. Next (right), we zoom in and draw a box around this emission. We have also at this stage dismissed the tape deck and Position Tracking parts of the display (§ 7.2.1) as they are not used here. We have also changed the <code>iterations</code> to 30 for this boxed clean. We will now hit the Next Action Continue Cleaning button (the green clockwise arrow) to start cleaning.	239
5.3	We continue in our interactive <code>cleaning</code> of Jupiter from where Figure 5.2 left off. In the first (left) panel, we have cleaned 30 iterations in the region previously marked, and are zoomed in again ready to extend the mask to pick up the newly revealed emission. Next (right), we have used the Polygon tool to redraw the mask around the emission, and are ready to Continue Cleaning for another 100 iterations. . . .	240

5.4	We continue in our interactive <code>clean</code> ing of Jupiter from where Figure 5.3 left off. In the first (left) panel, it has cleaned deeper, and we come back and zoom in to see that our current mask is good and we should clean further. We change <code>npercycle</code> to 500 (from 100) in the box at upper right of the window. In the final panel (right), we see the results after this clean. The residuals are such that we should terminate the <code>clean</code> using the red X button and use our model for self-calibration.	241
5.5	After <code>clean</code> and self-calibration using the intensity image, we arrive at the final polarization image of Jupiter. Shown in the <code>viewer</code> superimposed on the intensity raster is the linear polarization intensity (green contours) and linear polarization B-vectors (vectors). The color of the contours and the sampling and rotation by 90 degrees of the vectors was set in the Display Options panel. A LEL expression was used in the Load Data panel to mask the vectors on the polarized intensity.	242
5.6	Screen-shot of the interactive <code>clean</code> window during deconvolution of the NGC5921 spectral line dataset. Note where we have selected the mask to apply to the <code>Displayed Plane</code> rather than <code>All Channels</code> . We have just used the Polygon tool to draw a mask region around the emission in this channel, which will apply to this channel only.	243
6.1	NGC2403 VLA moment zero (left) and NGC4826 BIMA moment one (right) images as shown in the <code>viewer</code>	277
7.1	The Viewer Display Panel (left) and Data Display Options (right) panels that appear when the <code>viewer</code> is called with the image cube from NGC5921 (<code>viewer('ngc5921.usecase.clean.i</code> The initial display is of the first channel of the cube.	290
7.2	The Viewer Display Panel (left) and Data Display Options (right) panels that appear when the <code>viewer</code> is called with the NGC5921 Measurement Set (<code>viewer('ngc5921.usecase.ms')</code>).2	
7.3	The display panel's Main Toolbar appears directly below the menus and contains 'shortcut' buttons for most of the frequently-used menu items.	294
7.4	The 'Mouse Tool' Bar allows you to assign separate mouse buttons to tools you control with the mouse within the image display area. Initially, zooming, color adjustment, and rectangular regions are assigned to the left, middle and right mouse buttons, respectively.	294
7.5	The Load Data - Viewer panel that appears if you open the <code>viewer</code> without any <code>infile</code> specified, or if you use the <code>Data:Open</code> menu or Open icon. You can see the images and MS available in your current directory, and the options for loading them. 298	
7.6	The Load Data - Viewer panel as it appears if you select an image. You can see all options are available to load the image as a Raster Image , Contour Map , Vector Map , or Marker Map . In this example, clicking on the Raster Image button would bring up the displays shown in Figure 7.1.	300
7.7	The Basic Settings category of the Data Display Options panel as it appears if you load the image as a Raster Image . This is a zoom-in for the data displayed in Figure 7.1.	302
7.8	Example curves for scaling power cycles.	303

7.9	The Viewer Display Panel (left) and Data Display Options panel (right) after choosing Contour Map from the Load Data panel. The image shown is for channel 11 of the NGC5921 cube, selected using the Animator tape deck, and zoomed in using the tool bar icon. Note the different options in the open Basic Settings category of the Data Display Options panel.	305
7.10	The Viewer Display Panel (left) and Data Display Options panel (right) after overlaying a Contour Map of velocity on a Raster Image of intensity. The image shown is for the moments of the NGC5921 cube, zoomed in using the tool bar icon. The tab for the contour plot is open in the Data Display Options panel.	306
7.11	The Image Profile panel that appears if you use the Tools:Spectral Profile menu, and then use the rectangle or polygon tool to select a region in the image. You can also use the crosshair to get the profile at a single position in the image. The profile will change to track movements of the region or crosshair if moved by dragging with the mouse.	307
7.12	Selecting an image region with the polygon tool.	308
7.13	A typical pop-up that appears for the selection of regions and boxes	309
7.14	A multi-panel display set up through the Viewer Canvas Manager	310
7.15	The Load Data - Viewer panel as it appears if you select an MS. The only option available is to load this as a Raster Image . In this example, clicking on the Raster Image button would bring up the displays shown in Figure 7.2.	311
7.16	The MS for NGC4826 BIMA observations has been loaded into the viewer. We see the first of the spw in the Display Panel , and have opened up MS and Visibility Selections in the Data Display Options panel. The display panel raster is not full of visibilities because spw 0 is continuum and was only observed for the first few scans. This is a case where the different spectral windows have different numbers of channels also.	313
7.17	The MS for NGC4826 from Figure 7.16, now with the Display Axes open in the Data Display Options panel. By default, channels are on the Animation Axis and thus in the tapedeck, while spectral window and polarization are on the Display Axes sliders.	315
7.18	The MS for NGC4826, continuing from Figure 7.17. We have now put spectral window on the Animation Axis and used the tapedeck to step to spw 2 , where we see the data from the rest of the scans. Now channels is on a Display Axes slider, which has been dragged to show Channel 33	316
7.19	Setting up to print to a file. The background color has been set to white , the line width to 2, and the print resolution to 600 dpi (for an postscript plot). To make the plot, use the Save button on the Viewer Print Manager panel (positioned in the figure in the upper right) and select a format with the drop-down, or use the Print button to send directly to a printer.	320
A.1	Wiring diagram for the SDtask sdcal . The stages of processing within the task are shown, along with the parameters that control them.	327

A.2	Total power data display using <code>sdtpimaging</code> , with <code>calmode='baseline'</code> . The top panel shows uncalibrated data versus row numbers. The middle panel shows baseline fitting of each scan (only shown here the last scan). The bottom panel shows the calibrated (baseline subtracted) data.	384
A.3	Multi-panel display of the scantable. There are two plots per scan indicating the <code>_psr</code> (reference position data) and the <code>_ps</code> (source data).	403
A.4	Two panel plot of the calibrated spectra. The GBT data have a separate scan for the SOURCE and REFERENCE positions so scans 20,21,22 and 23 result in these two spectra.	404
A.5	Calibrated spectrum with a line at zero (using histograms).	404
A.6	FLS3a HI emission. The display illustrates the visualization of the data cube (left) and the profile display of the cube at the cursor location (right); the Tools menu of the Viewer Display Panel has a Spectral Profile button which brings up this display. By default, it grabs the left-mouse button. Pressing down the button and moving in the display will show the profile variations.	407

Chapter 1

Introduction

This document describes how to calibrate and image interferometric and single-dish radio astronomical data using the CASA (Common Astronomy Software Application) package. CASA is a suite of astronomical data reduction tools and tasks that can be run via the IPython interface to Python. CASA is being developed in order to fulfill the data post-processing requirements of the ALMA and EVLA projects, but also provides basic and advanced capabilities useful for the analysis of data from other radio, millimeter, and submillimeter telescopes.

You have in your hands the latest release of CASA. This package is under active development, and thus there are a number of caveats and limitations for the use of this package. See the release notes (§ 1.1) below for more information, and pay heed to the numerous **ALERT**s placed throughout this reference. You can expect regular updates and patches, as well as increasing functionality.

This user reference and cookbook is a task-based walk-through of interferometric data reduction and analysis. In CASA, **tasks** represent the more streamlined operations that a typical user would carry out. The idea for having tasks is that they are simple to use, provide a more familiar interface, and are easy to learn for most astronomers who are familiar with radio interferometric data reduction (and hopefully for novice users as well). In CASA, the **tools** provide the full capability of the package, and are the atomic functions that form the basis of data reduction. These tools augment the tasks, or fill in gaps left by tasks that are under development but not yet available. See the **CASA Toolkit Manual** for more details on the tools (available from casa.nrao.edu). Note that in most cases, the tasks are Python interface scripts to the tools, but with specific, limited access to them and a standardized interface for parameter setting. The tasks and tools can be used together to carry out more advanced data reduction operations.

Inside the Toolkit:

Throughout this user reference, we will occasionally intersperse boxed-off pointers to parts of the toolkit that power users might want to explore.

For the moment, the audience is assumed to have some basic grasp of the fundamentals of synthesis imaging, so details of how a radio interferometer or telescope works and why the data needs to undergo calibration in order to make synthesis images are left to other documentation — a good

place to start might be *Synthesis Imaging in Radio Astronomy II* (1999, ASP Conference Series Vol. 180, eds. Taylor, Carilli & Perley).

This reference is broken down by the main phases of data analysis:

- data import, export, and selection (Chapter 2),
- examination and flagging of data (Chapter 3),
- interferometric calibration (Chapter 4),
- interferometric imaging (Chapter 5),
- image analysis (Chapter 6), and
- data and image visualization (Chapter 7).

There are also special chapters in the Appendix on

- single dish data analysis (Chapter A), and
- simulation (Chapter B).

These are included for users that will be doing EVLA and ALMA telescope commissioning and software development.

The general appendices provide more details on what's happening under the hood of CASA, as well as supplementary material on tasks, scripts, and relating CASA to other packages. These appendices include:

- obtaining and installing CASA (Appendix C),
- more details about Python and CASA (Appendix D),
- a discussion of the Hamaker-Bregman-Sault Measurement Equation (Appendix E),
- annotated scripts for typical data reduction cases (Appendix F), and
- CASA dictionaries to AIPS, MIRIAD, and CLIC (Appendix G).
- Writing your own CASA Task (Appendix H).

The CASA User Documentation includes:

- **CASA User Reference & Cookbook** — this document, a task-based data analysis walk-through and instructions;
- **CASA in-line help** — accessed using `help` in the `casapy` interface;

- The **CASA Toolkit Reference Manual** — details on a specific task or tool does and how to use it.
- The **CASA Task Reference Manual** — the information from the inline help and task documentation, available online in HTML.

The CASA home page can be found at:

`http://casa.nrao.edu`

From there you can find documentation and assistance for the use of the package, including the User Documentation. You will also find information on how to obtain the latest release and receive user support.

There is also a CASAGuides Wiki

`http://casaguides.nrao.edu`

that contains helpful information on CASA startup, AIPS-to-CVASA cheatsheet, example scripts on processing your data in CASA, along with hints and tricks to best use this package.

1.1 About This Release

This release (3.0.1) is a patch to the 3.0.0 release. The patch is mainly driven by the first light of the EVLA. CASA, however, is still under development, and some features might change in future releases. This should be taken into account as users begin to learn the package. We will do our best to point out commands, tasks, and parameters that are likely to change underfoot. Although many bugs were eradicated in the Beta Release stage, there are some that persist. We will do our best to stamp these out as soon as we find them. See the release notes for the current version for more details. In this reference, we will try to point out known pitfalls and workarounds in the **ALERT** boxes or notes in the text.

ALERT

Boxes like this will bring to your attention some of the features (or lack thereof) in the current release of CASA. There are also **ALERT** notes in the text.

We occasionally issue patches to the current release in between formal releases. Please check the CASA Home page (`http://casa.nrao.edu`) regularly to look for updates to the release and the documentation, and to check the list of known problems. You can find the contact information for feedback here also.

1.1.1 What's New in Release 3.0.1

Release 3.0.1 is mainly intended to be a patch to the previous 3.0.0 release of CASA. It is timed to coincide with the first scientific data delivery from the EVLA. However, many improvements to the code have been made and a number of bugs have been fixed. The improvements include:

- **Flagging**

- up to 30% speed improvements in `flagdata`, in particular for data with many spectral windows

- **Imaging**

- new task `csvclean` to support ALMA commissioning. This task does an invert of the visibilities and deconvolve in the image plane. This task does not do a uvdata subtraction (aka Cotton-Schwab major cycle) of model visibility as in `clean`
- `clean`: 'chaniter' is now separated from the 'interactive' parameter and available for 'channel', 'frequency' and 'velocity' modes. Cleaning sequentially by channel is now available not just for interactive but also for non-interactive `clean`

- **Data examination/editing**

- `ms.msseltoindex` now returns list of channels only those channels selected by the "spw" parameter. It also now returns the baseline/antenna selection.
- `listvis` can read `FLOAT_DATA` columns from single dish MS. Also: improved listing of auto-correlation MS on a Mac
- In `plotms`, improved "locate" output format using field and antenna names, instead of indices
- In `plotms`, new axes choices supported (when not averaging): `Wt`, `ImWt`
- `plotms` will plot caltables. NB: This is still under development and should be used with caution. Many gui options—e.g., averaging, transformations, some axis options—are not supported, even though they appear to be available.

- **Calibration**

- new parameter 'minsnr' in `bandpass` for channel based, 'B', solutions. 'minsnr' defines the signal to noise threshold for a solution to be valid
- The EPJones table now has separate pointing offsets for the two polarizations
- New gaincurves for the EVLA, and some bugs fixed in gaincurve application
- Channelized R-L phase calibration now supported in `polcal` via `poltype='Xf'`.
- Provisional "Perley-Butler 2010" flux density scale now supported in `setjy`. NB: the default is still "Perley-Taylor 1999", until the authors complete their analysis. "Perley-Butler 2010" is recommended for high VLA/EVLA frequencies (> 15 GHz) as it is bootstrapped to the absolute flux density calibration of WMAP using planets at the higher frequencies.

- **Data and Image Manipulation**

- the default interpolation method in spectral gridding within `clean` and `cvel` is now set to 'linear' instead of 'nearest'
- `immath`: mode 'pola' and 'poli' calculations now can be done if a single, multi-stokes image is provided; added linear polarization intensity threshold for 'pola'

- **Simulation**

- introducing a new task interface `simdata2`: modular / independent execution of sub-tasks: modify sky model, calculate mosaic pointings, predict visibilities, (re)image the data, and analyze the simulated image
- `simdata2` incorporates single dish capability from `sdsim`, including joint deconvolution of simulated total power and interferometric data
- the old `simdata` and `sdsim` tasks are preserved for backward capability, with some bugs fixed.

- **Data im/export**

- `exportuvfits` now has a parameter `padwithflags` which allows users to add flags to missing data sections if `combinespw` is set. With both parameters set, `exportuvfits` will export the spectral windows as IFs
- `importevla` task has now an option to write autocorrelation and/or crosscorrelation data to an MS. The task can also flag all data that come as zeros

- **Single Dish**

- header, panel layout, and legend location control for `sdplot`. Also a new parameter `'plotstyle'` is added to allow more flexibility in arranging plots.
- `sdsim` has new options `'checkinputs'`, `'fidelity'`, and `'display'` for improved plotting flexibility.
- `asaplotter` has now options to change the panel layout
- `msmoments` now also works on single-dish measurement sets. Added subparameters for data selection; `'includerow'` and `'excluderow'` were renamed to `'includemask'` and `'excludemask'`

- **Other**

- up to 2.5 times speed improvement in the creation of scratch columns (`'model'` and `'corrected_data'` columns)
- `'rm -r'` within CASA is now calling `shutil.rmtree` for cleaner removal of datasets and tables

- **Flagging**

- flagging statistics, `flagdata(mode='summary')` now returns statistical information
- possible to query / filter the output of `flagdata(mode='summary')`
- Fixed/implemented inrow selections for `mode='summary'`

- **Imaging**

- re-implementation of `boxit` to write boxes in ascii format to be handled by `viewer` and `clean`

- Improved handling of sidelobes of the psf
- default padding of 1.2 to avoid edge effects in deconvolution
- **Data examination/editing**
 - significant `plotms` improvements in overall usability
 - `plotms` can now relabel the visibilities to velocities and a variety of velocity and frequency frames and velocity definitions can be chosen
 - `plotms` can perform shifts of the phase center
 - new parameters in `varlist` to work with expressions
 - improved handling of spws with different numbers of channels
 - updates region treatment in `viewer` (note that there may be further changes in the future)
- **Calibration**
 - improvements to leakage term solution
 - improved handling and computing of uvw coordinates
 - handling of aatm libraries later than v.0.08
 - updating ALMA calibration algorithms
- **Data and Image Manipulation**
 - improved `split` to include averaging in spectral and time domain simultaneously
 - `split` can average MSes with differently shaped spwd.
 - single dish support in `split`
 - `split` retains the weather table
 - add position angle to `imsmooth` and add option to provide either smoothing or smoothed beam
 - cross-spw solution for continuum `uvcontsub2` subtraction
 - `uvcontsub2` now makes a new MS instead of altering the input one. This also makes the continuum estimate easier to use.
 - easy coordinate transformation options (e.g. J2000 to B1950 or Galactic) in `imregrid`
 - improved regridding of velocities into different frames using `cvel`. Now in line with regridding in `clean`
- **Simulation**
 - Import of multiple pointings via external text file
 - `mosaicsize` parameter is now used for specifying the angular size of sky to observe, independently of the user's choices of cell and `imsize` in `simdata`
 - new `simdata` option to add a calibrator observation interleaved with the science target mosaic has been added as an experimental feature. Please report any issues or questions.

- transformation of input model image cubes into the 4-d format used internally (direction, stokes, spectral) in `simdata` is now more robust to different choices of direction and spectral coordinates than previously
 - parameter name changes in `simdata` and `sdsim`
 - improved single dish simulations
- **Data im/export**
 - single dish parameter in `importasdm`
 - im/export of weather data and flags stored in the ASDM
 - fixed spw setting of ALMA autocorrelation data
 - `exportasdm` - version 1 of a new task to turn an arbitrary MS into an ASDM. This is mainly geared towards ALMA testing and to store simulated data in the ALMA archive. In version 1 all relevant tables except the weather and the history table are filled.
- **Single Dish**
 - use of `SRCTYPE` as identifier for scan intent (ON, OFF, etc)
 - allow arithmetic operations on scantables
 - antenna selection for MS data with multiple antennas
 - improved plotting of sd data
- **Documentation**
 - EVLA examples on `casaguides.nrao.edu`
 - rationalizing of documentation into cookbook and `casaguides.nrao.edu`
 - `plotms` description in cookbook
- **Other**
 - preparations for code generalizations to implement parallel computing on clusters
 - TM library in CASA is updated to `aatm-0.3.0`. A method in atmosphere tools, `atmosphere.getAtmTypeHF` is removed due to the upgrade
 - The default I/O tile size was reduced to 32 KB. This speeds up some I/O modes, as used by flagging and `exportuvfits`.

1.2 CASA Basics — Information for First-Time Users

This section assumes that CASA has been installed on your LINUX or OSX system. See Appendix C for instructions on how to obtain and install CASA.

1.2.1 Before Starting CASA

First, you will most likely be starting CASA running from a working directory that has your data in it, or at least where you want your output to go. It is easiest to start from there rather than changing directories inside `casapy`. **ALERT:** There is at least one task (`plotxy`) that fails if the path to your working directory contains spaces in its name, e.g. `/users/smyers/MyTest/` is fine, but `/users/smyers/My Test/` is not! We will try to bulletproof the file-handling better.

If you have done a default installation under Linux using rpms, or on the Mac with the CASA application, then there should be a `sh` script called `casapy` in the `/usr/bin` area which is in your path. This shell will set up its environment and run the version of `casapy` that it points to. If this is how you set up the system, then you need to nothing further and can run `casapy`.

Depending on your setup, there may be other specially built versions available. For example at the NRAO AOC, the “stable build” can be started by running “`casapy-test`”, e.g.

```
/usr/bin/casapy-test
```

On some systems, particularly if you have multiple versions installed, to define environment variables and the `casapy` alias, you will need to run one of the `casainit` shell scripts. The location of the startup scripts for CASA will depend upon where you installed CASA on your system. For a default installation this will likely be in `/usr/bin/casapy/`. For example, at the NRAO AOC, the current release is executed as

```
/usr/bin/casapy
```

and uses the pathname to

```
/usr/lib/casapy/23.0.6654-001
```

Sometimes, you will have multiple non-default versions (for example, various development versions). For example at the NRAO AOC, the “stable build” is in `/home/casa`. Then, to use this version:

```
In bash:
> . /home/casa/casainit.sh
or for csh:
> source /home/casa/casainit.csh
```

depending on what shell you are running (Bourne or [t]csh).

ALERT: If you want to run the `casabrowser` (see § 3.6) outside of the `casapy` shell, then you will need to put the CASA root in your path using one of the above mechanisms, or give its path explicitly.

1.2.1.1 Environment Variables

Before starting up `casapy`, you should set or reset any *environment variables* needed, as CASA will adopt these on startup. For example, the `PAGER` environment variable determines how help is displayed in the CASA terminal window (see § 1.2.8.3). The choices are `less`, `more`, and `cat`.

In `bash`, pick one of

```
PAGER=less
PAGER=more
PAGER=cat
```

followed by

```
export PAGER
```

In `csh` or `tcsh`, pick one of

```
setenv PAGER less
setenv PAGER more
setenv PAGER cat
```

The actions of these are as if you were using the equivalent Unix shell command to view the help material. See § 1.2.8.3 for more information on these choices. We recommend using the `cat` option for most users, as this works smoothly both interactively and in scripts.

1.2.1.2 Where is CASA?

Note that the path to the CASA installation, which contains the scripts and data repository, will also depend upon the installation. With a default installation under Linux this will probably be in

```
/usr/lib/casapy/
```

while in a Mac OSX default install it will likely be an application (in the Applications folder), with the data repository in:

```
/opt/casa/
```

You can find the location after initialized by looking at the `CASAPATH` environment variable. You can find it within `casapy` by

```
pathname=os.environ.get('CASAPATH').split()[0]
print pathname
```

1.2.2 Starting CASA

After having run the appropriate `casainit` script, CASA is started by typing

```
casapy
```

on the UNIX command line, e.g.

```
casapy
```

After startup information, you should get an IPython

```
CASA <1>:
```

command prompt in the xterm window where you started CASA. CASA will take approximately 10 seconds to initialize at startup in a new working directory; subsequent startups are faster. CASA is active when you get a

```
CASA <1>
```

prompt in the command line interface. You will also see a `logger` GUI appear on your Desktop (usually near the upper left). *Note: Under MacOSX the logger will appear in a Console window.*

You also have the option of starting CASA with various `logger` options (see § 1.4.2.1). For example, if you are running remotely in a terminal window without an X11 connection, or if you just do not want to see the logger GUI, and want the `logger` messages to come to your terminal, do

```
casapy --nologger --log2term
```

See § 1.4.2 for information on the `logger` in general.

1.2.3 Ending CASA

You can exit CASA by typing `quit`. This will bring up the query

```
Do you really want to exit ([y]/n)?
```

to give you a chance in case you did not mean to exit. You can also quit using `%exit` or `CTRL-D`.

If you don't want to see the question "Do you really want to exit [y]/n?", then just type `Exit` or `exit` and CASA will stop right then and there.

1.2.4 What happens if something goes wrong?

ALERT: Please check the CASA Home Page for Release Notes and FAQ information including a list of known problems. If you think you have encountered an unknown problem, please consult the CASA HelpDesk (contact information on the CASA Home Page). See also the caveats to this Release (§ 1.1 for pointers to our policy on User Support).

First, always check that your inputs are correct; use the

```
help <taskname>
```

(§ 1.2.8.2) or

```
help par.<parameter name>
```

(§ 1.2.8.4) to review the inputs/output.

1.2.5 Aborting CASA execution

If something has gone wrong and you want to stop what is executing, then typing `CNTL-C` (Control and C keys simultaneously) will usually cleanly abort the application. This will work if you are running a task synchronously. If this does not work on your system, or you are running a task asynchronously (§ 1.3.4) then try `CNTL-Z` to put the task or shell in the background, and then follow up with a `kill -9 <PID>` where you have found the relevant `casapy` process ID (PID) using `ps` (see § 1.2.6 below).

See § 1.3.2 for more information on running tasks.

If the problem causes CASA to crash, see the next sub-section.

1.2.6 What happens if CASA crashes?

Usually, restarting `casapy` is sufficient to get you going again after a crash takes you out of the Python interface. Note that there may be spawned subprocesses still running, such as the `casaviewer` or the `logger`. These can be dismissed manually in the usual manner. After a crash, there may also be hidden processes. You can find these by listing processes, e.g. in linux:

```
ps -elf | grep casa
```

or on MacOSX (or other BSD Unix):

```
ps -aux | grep casa
```

You can then kill these, for example using the Unix `kill` or `killall` commands. This may be necessary if you are running remotely using `ssh`, as you cannot logout until all your background processes are terminated. For example,

```
killall ipcontroller
```

or

```
killall Python
```

will terminate the most common post-crash zombies.

1.2.7 Python Basics for CASA

Within CASA, you use Python to interact with the system. This does not mean an extensive Python course is necessary - basic interaction with the system (assigning parameters, running tasks) is straightforward. At the same time, the full potential of Python is at the more experienced user's disposal. Some further details about Python, IPython, and the interaction between Python and CASA can be found in Appendix D.

The following are some examples of helpful hints and tricks on making Python work for you in CASA.

1.2.7.1 Variables

Python variables are set using the `<parameter> = <value>` syntax. Python assigns the type dynamically as you set the value, and thus you can easily give it a non-sensical value, e.g.

```
vis = 'ngc5921.ms'
vis = 1
```

The CASA parameter system will check types when you run a task or tool, or more helpfully when you set inputs using `inp` (see below). CASA will check and protect the assignments of the global parameters in its namespace.

Note that Python variable names are case-sensitive:

```
CASA <109>: Foo = 'bar'
CASA <110>: foo = 'Bar'
CASA <111>: foo
  Out[111]: 'Bar'
CASA <112>: Foo
  Out[112]: 'bar'
```

so be careful.

Also note that mis-spelling a variable assignment will not be noticed (as long as it is a valid Python variable name) by the interface. For example, if you wish to set `correlation='RR'` but instead type `corellation='RR'` you will find `correlation` unset and a new `corellation` variable set. Command completion (see §1.2.8.1) should help you avoid this.

1.2.7.2 Lists and Ranges

Sometimes, you need to give a task a list of indices. If these are consecutive, you can use the Python `range` function to generate this list:

```
CASA <1>: iflist=range(4,8)
CASA <2>: print iflist
```



```
[4, 5, 6, 7]
CASA <3>: iflist=range(4)
CASA <4>: print iflist
[0, 1, 2, 3]
```

See Appendix D.3 for more information.

1.2.7.3 Indexes

As in C, Python indices are 0-based. For example, the first element in a list `antlist` would be `antlist[0]`:

```
CASA <113>: antlist=range(5)
CASA <114>: antlist
  Out[114]: [0, 1, 2, 3, 4]
CASA <115>: antlist[0]
  Out[115]: 0
CASA <116>: antlist[4]
  Out[116]: 4
```

CASA also uses 0-based indexing internally for elements in the Measurement Set (MS – the basic construct that contains visibility and/or single dish data; see Chapter 2). Thus, we will often talk about Field or Antenna “ID”s which will start at 0. For example, the first field in an MS would have `FIELD.ID==0` in the `MSselect` syntax, and can be addressed as be indexed as `field='0'` in most tasks, as well as by name `field='0137+331'` (assuming that's the name of the first field). You will see these indices in the MS summary from the task `listobs`.

1.2.7.4 Indentation

Python pays attention to the indentation of lines, as it uses indentation to determine the level of nesting in loops. Be careful when cutting and pasting: if you get the wrong indentation, then unpredictable things can happen (usually it just gives an error).

See Appendix D.2 for more information.

1.2.7.5 System shell access

If you want to access system commands from a script, use the `os.system` command (Appendix D.6.1).

In interactive mode, any input line beginning with a `'!'` character is passed verbatim (minus the `'!'`, of course) to the underlying operating system. Also, several common commands (`ls`, `pwd`, `less`) may be executed with or without the `'!'`, although the `cp` command must use `'!'` and `cd` must be executed without the `'!'`. For example:

```
CASA <5>: !rm -r mydata.ms
```

Note that if you want to access a Unix environment variable, you will need to prefix with a double `$$` instead of a single `$` — for example, to print the value of the `$PAGER` variable, you would use

```
CASA <6>: !echo $$PAGER
```

See Appendix D.6 for more information.

1.2.7.6 Executing Python scripts

You can execute Python scripts (ASCII text files containing Python or `casapy` commands) using the `execfile` command. For example, to execute the script contained in the file `myscript.py` (in the current directory), you would type

```
CASA <7>: execfile('myscript.py')
```

or

```
CASA <8>: execfile 'myscript.py'
```

which will invoke the IPython auto-parenthesis feature.

NOTE: in some cases, you can use the IPython `run` command instead, e.g.

```
CASA <9>: run myscript.py
```

In this case, you do not need the quotes around the filename. This is most useful for re-initializing the task parameters, e.g.

```
CASA <10>: run clean.last
```

(see § 1.3.5.8).

See Appendix D.11 for more information.

1.2.8 Getting Help in CASA

1.2.8.1 TAB key

At **any** time, hitting the `<TAB>` key will complete any available commands or variable names and show you a list of the possible completions if there's no unambiguous result. It will also complete filenames in the current directory if no `CASA` or Python names match.

For example, it can be used to list the available functionality using minimum match; once you have typed enough characters to make the command unique, `<TAB>` will complete it.

```

CASA <15>: cle<TAB>
clean                clean_description      clearcal_check_params
clearplot            clearstat
clean_check_params  clear                clearcal_defaults
clearplot_defaults  clearstat_defaults
clean_defaults      clearcal              clearcal_description
clearplot_description clearstat_description

```

1.2.8.2 help <taskname>

Basic information on an application, including the parameters used and their defaults, can be obtained by typing `pdoc task`, `help task`, `help 'task'` or `task?`. The `pdoc task` currently gives the cleanest documentation format with the smallest amount of object-oriented (programmer) output. This inline help provides a one line description of the task and then lists all parameters, a brief description of the parameter, the parameter default, an example setting the parameter and any options if there are limited allowed values for the parameter.

For example:

```
Import VLA archive file(s) to a measurement set
```

```
Imports an arbitrary number of VLA archive-format data sets into
a casa measurement set.  If more than one band is present, they
will be put in the same measurement set but in a separate spectral
window.  The task will handle old style and new style VLA (after
July 2007) archive data and apply the tsys to the data and to
the weights.
```

Keyword arguments:

```
archivefiles -- Name of input VLA archive file(s)
               default: none.  Must be supplied
               example: archivefiles = 'AP314_A959519.xp1'
               example: archivefiles=['AP314_A950519.xp1', 'AP314_A950519.xp2']
```

```
vis -- Name of output visibility file
      default: none.  Must be supplied.
      example: vis='NGC7538.ms'
      Will not over-write existing ms of same name.
      A backup flag-file version 'Original' will be made in
      vis.flagversions.  See help flagmanager
```

```
bandname -- VLA Frequency band
           default: => '' = all bands
           example: bandname='K'
           Options: '4'=48-96 MHz, 'P'=298-345 MHz, 'L'=1.15-1.75 GHz,
           'C'=4.2-5.1 GHz, 'X'=6.8-9.6 GHz, 'U'=13.5-16.3 GHz,
           'K'=20.8-25.8 GHz, 'Q'=38-51 GHz
```

```
frequencytol -- Tolerance in frequency shift in making spectral windows
               default: => 150000 (Hz).  For Doppler shifted data, <10000 Hz may
               may produce too many unnecessary spectral windows.
               example: frequencytol = 1500000.0 (units = Hz)
```

```

project -- Project name to import from archive files:
  default: '' => all projects in file
  example: project='AL519'
  project = 'al519' or AL519 will work. Do not include
  leading zeros; project = 'AL0519' will not work.
starttime -- Time after which data will be considered for importing
  default: '' => all: Date must be included.
  syntax: starttime = '2003/1/31/05:05:23'
stoptime -- Time before which data will be considered for importing
  default: '' => all: Date must be included.
  syntax: stoptime = '2003/1/31/08:05:23'
applysys -- Apply data scaling and weight scaling by nominal
  sensitivity (~Tsys)
  default: True. Strongly recommended
autocorr -- import autocorrelations to ms
  default: => False (no autocorrelations)
antnamescheme -- 'old' or 'new' antenna names.
  default => 'new' gives antnenna names
  'VA04' or 'EA13 for VLA telescope 04 and 13 (EVLA)
  'old' gives names '04' or '13'
keepblanks -- Should sources with blank names be filled into the data base
  default => false. Do not fill
  These scans are tipping scans (as of June 1, 2009) and should not
  be filled in the visibility data set.
evlabands -- Use the EVLA's center frequency and bandwidths for frequencies
  specified via wavelength or band.
  default => True.
async -- Run asynchronously
  default = False; do not run asychronously

```

You can also get the short help for a CASA tool method by typing 'help tool.method'.

```
CASA <46>: help ia.subimage
```

```
Help on built-in function subimage:
```

```
subimage(...)
```

```
Create a (sub)image from a region of the image
```

```
----- Parameters -----
```

```

outfile: Output image file name. Default is unset.
region:  Region of interest. Default is whole image.
mask:    OTF mask, Boolean LEL expression or mask region. Default is none.
dropdeg: Drop degenerate axes false
overwrite: Overwrite (unprompted) pre-existing output file? false
list:    List informative messages to the logger true

```

```
-----
```

```

outfile
region
mask
dropdeg  = false
overwrite = false

```

```
list      = true
-----
```

For a full list of keywords associated with the various tools, see the **CASA User Reference Manual**.

1.2.8.3 help and PAGER

Your `PAGER` environment variable (§ 1.2.1) determines how help is displayed in the terminal window where you start CASA. If you set your `bash` environment variable `PAGER=less` (`setenv PAGER less` in `csh`) then typing `help <taskname>` will show you the help but the text will vanish and return you to the command line when you are done viewing it. Setting `PAGER=more` (`setenv PAGER more`) will scroll the help onto your command window and then return you to your prompt (but leaving it on display). Setting `PAGER=cat` (`setenv PAGER cat`) will give you the `more` equivalent without some extra formatting baggage and is the recommended choice.

If you have set `PAGER=more` or `PAGER=less`, the `help` display will be fine, but the display of `'taskname?'` will often have confusing formatting content at the beginning (lots of `ESC` surrounding the text). This can be remedied by exiting `casapy` and doing an `'unset PAGER'` (`unsetenv PAGER` in `[t]csh`) at the Unix command line.

You can see the current value of the `PAGER` environment variable with CASA by typing:

```
!echo $$PAGER
```

(note the double `$$`). This will show what command paging is pointed to.

1.2.8.4 help par.<parameter>

Typing `help par.<parameter>` provides a brief description of a given parameter `<parameter>`.

```
CASA <46>: help par.robust
Help on function robust in module parameter_dictionary:

robust()
  Brigg's robustness parameter.

  Options: -2.0 (close to uniform) to 2.0 (close to natural)
```

1.2.8.5 Python help

Typing `help` at the `casapy` prompt with no arguments will bring up the native Python help facility, and give you the `help>` prompt for further information; hitting `<RETURN>` at the help prompt returns you to the CASA prompt.

```
CASA <2>: help
-----> help()
```

Welcome to Python 2.5! This is the online help utility.

If this is your first time using Python, you should definitely check out the tutorial on the Internet at <http://www.python.org/doc/tut/>.

Enter the name of any module, keyword, or topic to get help on writing Python programs and using Python modules. To quit this help utility and return to the interpreter, just type "quit".

To get a list of available modules, keywords, or topics, type "modules", "keywords", or "topics". Each module also comes with a one-line summary of what it does; to list the modules whose summaries contain a given word such as "spam", type "modules spam".

```
help> keywords
```

Here is a list of the Python keywords. Enter any keyword to get more help.

and	else	import	raise
assert	except	in	return
break	exec	is	try
class	finally	lambda	while
continue	for	not	yield
def	from	or	
del	global	pass	
elif	if	print	

```
help>
```

```
# hit <RETURN> to return to CASA prompt
```

You are now leaving help and returning to the Python interpreter. If you want to ask for help on a particular object directly from the interpreter, you can type "help(object)". Executing "help('string')" has the same effect as typing a particular string at the help> prompt.

Further help in working within the Python shell is given in Appendix D.

1.3 Tasks and Tools in CASA

Originally, CASA consisted of a collection of tools, combined in the so-called toolkit. Since the majority of prospective users is far more familiar with the concept of tasks, an effort is underway to replace most - if not all - toolkit functionality by tasks.

While running CASA, you will have access to and be interacting with tasks, either indirectly by providing parameters to a task, or directly by running a task. Each task has a well defined purpose, and a number of associated parameters, the values of which are to be supplied by the user. Technically speaking, tasks are built on top of tools - when you are running a task, you are running tools in the toolkit, though this should be transparent.

As more tasks are being written, and the functionality of each task is enhanced, there will be less and less reason to run tools in the toolkit. We are working toward a system in which direct access to the underlying toolkit is unnecessary for all standard data processing.

1.3.1 What Tasks are Available?

As mentioned in the introduction, tasks in CASA are python interfaces to the more basic toolkit. Tasks are executed to perform a single job, such as loading, plotting, flagging, calibrating, and imaging the data.

Basic information on tasks, including the parameters used and their defaults, can be obtained by typing `help <taskname>` or `<taskname>?` at the CASA prompt, where `<taskname>` is the name of a given task. As described above in § 1.2.8.2, `help <taskname>` provides a description of the task and then lists all parameters, a brief description of the parameter, the parameter default, an example setting the parameter and any options if there are limited allowed values for the parameter.

To see what tasks are available in CASA, use `tasklist`, e.g.

```
CASA <5>: tasklist
-----> tasklist()
Available tasks:
```

Import/Export	Information	Data Editing	Display/Plot
-----	-----	-----	-----
<code>importvla</code>	<code>listcal</code>	<code>(fixvis)</code>	<code>clearplot</code>
<code>(importasdm)</code>	<code>listhistory</code>	<code>flagautocorr</code>	<code>plotants</code>
<code>importfits</code>	<code>listobs</code>	<code>flagdata</code>	<code>plotcal</code>
<code>importuvfits</code>	<code>listvis</code>	<code>flagmanager</code>	<code>plotms</code>
<code>exportfits</code>	<code>imhead</code>	<code>plotms</code>	<code>plotxy</code>
<code>exportuvfits</code>	<code>imstat</code>	<code>plotxy</code>	<code>viewer</code>
	<code>vishead</code>		
	<code>visstat</code>		
Calibration	Imaging	Modelling	Utility
-----	-----	-----	-----
<code>accum</code>	<code>clean</code>	<code>setjy</code>	<code>help task</code>
<code>applycal</code>	<code>deconvolve</code>	<code>uvcontsub</code>	<code>help par.parameter</code>
<code>bandpass</code>	<code>feather</code>	<code>(uvcontsub2)</code>	<code>taskhelp</code>
<code>blcal</code>	<code>ft</code>	<code>uvmodelfit</code>	<code>tasklist</code>
<code>calstat</code>	<code>makemask</code>		<code>browsetable</code>
<code>gaincal</code>	<code>mosaic</code>		<code>clearplot</code>
<code>fluxscale</code>	<code>widefield</code>		<code>clearstat</code>

(fringecal)	concat
clearcal	filecatalog
listcal	startup
smoothcal	split
polcal	fixvis
hanningsmooth	

Image Analysis	Simulation	Single Dish (after running asap_init())
-----	-----	-----
imcontsub	simdata	sdaverage
imhead		sdbaseline
immath		sdcal
immoments		sdcoadd
imregrid		sdfit
imsmooth		sdfalg
imstat		sdimaging
imval		sdimprocess
(specfit)		sdlist
		sdmath
		sdplot
		sdsave
		sdscale
		(sdsim)
		sdsmooth
		sdstat
		sdtipimaging

User defined tasks

The tasks with name in parentheses are experimental. In the above case, the user has not defined any task him/herself.

Typing `taskhelp` provides a one line description of all available tasks.

```
CASA <6>: taskhelp
-----> taskhelp()
Available tasks:
```

```
accum          : Accumulate incremental calibration solutions into a calibration table
applycal      : Apply calibrations solutions(s) to data
autoclean     : CLEAN an image with automatically-chosen clean regions.
bandpass      : Calculates a bandpass calibration solution
blcal        : Calculate a baseline-based calibration solution (gain or bandpass)
boxit         : Box regions in image above given threshold value.
browsetable   : Browse a table (MS, calibration table, image)
calstat       : Displays statistical information on a calibration table
clean         : Invert and deconvolve images with selected algorithm
clearcal      : Re-initializes the calibration for a visibility data set
```



```

clearplot      : Clear the matplotlib plotter and all layers
clearstat     : Clear all autolock locks
concat        : Concatenate several visibility data sets.
cvel          : regrid an MS to a new spectral window / channel structure or frame
deconvolve    : Image based deconvolver
exportfits    : Convert a CASA image to a FITS file
exportuvfits  : Convert a CASA visibility data set to a UVFITS file:
feather       : Combine two images using their Fourier transforms
find          : Find string in tasks, task names, parameter names:
fixvis        : Recalculates (u, v, w)
flagautocorr  : Flag autocorrelations
flagdata      : All purpose flagging task based on selections
flagmanager   : Enable list, save, restore, delete and rename flag version files.
fluxscale     : Bootstrap the flux density scale from standard calibrators
fringecal     : Calculate a baseline-based fringe-fitting solution (phase, delay, delay-rate)
ft            : Insert a source model into the MODEL_DATA column of a visibility set:
gaincal       : Determine temporal gains from calibrator observations
gencal        : Specify Calibration Values of Various Types
hanningsmooth : Hanning smooth frequency channel data to remove Gibbs ringing
imcontsub     : Subtracts specified continuum channels from a spectral line data set
imfit         : Fit one or more elliptical Gaussian components on an image region(s)
imhead        : List, get and put image header parameters
immath        : Perform math operations on images
immoments     : Compute moments from an image
importasdm    : Convert an ALMA Science Data Model observation into a CASA visibility file
importfits    : Convert an image FITS file into a CASA image
importgmt      : Convert a UVFITS file to a CASA visibility data set
importoldasdm : Convert an ALMA Science Data Model observation into a CASA visibility file
importuvfits  : Convert a UVFITS file to a CASA visibility data set
importvla     : Import VLA archive file(s) to a measurement set
imregrid      : regrid an image onto a template image
imsmooth      : Smooth an image or portion of an image
imstat        : Displays statistical information from an image or image region
imval         : Get the data value(s) and/or mask value in an image.
listcal       : List antenna gain solutions
listhistory   : List the processing history of a dataset:
listobs       : List data set summary in the logger
listvis       : List measurement set visibilities.
makemask      : Make a mask image from rectangular boxes
mosaic        : Create a multi-field deconvolved image with selected algorithm
newflagdata   : Flag/Clip data based on selections
peel          : Do direction dependent selfcal(s) and optionally remove annoying sources.
plotants      : Plot the antenna distribution in the local reference frame:
plotcal       : An all-purpose plotter for calibration results
plotms        : A plotter/interactive flagger for visibility data.
plotxy        : An X-Y plotter/interactive flagger for visibility data.
polcal        : Determine instrumental polarization calibrations
rmtables      :
sdaverage     : ASAP SD task: do data selection, calibration, and averaging
sdbaseline    : ASAP SD task: fit/remove a spectral baseline
sdcal         : ASAP SD task: do sdaverage, sdsMOOTH, and sdbaseline in one task

```

```

sdcoadd      : ASAP SD task: coadd multiple scantables into one
sdfit       : ASAP SD task: fit a spectral line
sdflag      : ASAP SD spectral flagging task
sdimaging   : SD task: imaging for total power and spectral data
sdimprocess : SD task: task for single-dish image processing
sdlist      : ASAP SD task: list summary of single dish data
sdmath      : ASAP SD task: do simple arithmetic (subtraction, addition, multiplication, and division)
sdplot      : ASAP SD plotting task
sdsave      : ASAP SD task: save the sd spectra in various format
sdscale     : ASAP SD task: scale the sd spectra
sdsim       : single dish simulation task:
sdsmooth    : ASAP SD task: do smoothing of spectra
sdstat      : ASAP SD task: list statistics of spectral region
sdtipimaging : SD task: do a simple calibration and imaging for total power data
setjy       : Fills the model column with the visibility of a calibrator
simdata     : simulation task:
simdata2    : mosaic simulation task:
smoothcal   : Smooth calibration solution(s) derived from one or more sources:
specfit     : Fit 1-dimensional profile(s) or polynomial(s) to an image or image region
split       : Create a visibility subset from an existing visibility set
uvcontsub   : Continuum fitting and subtraction in the uv plane
uvcontsub2  : Continuum fitting and subtraction in the uv plane CALIBRATION VERSION
uvmodelfit  : Fit a single component source model to the uv data
uvsb        : Subtract/add model from/to the corrected visibility data.
viewer      : View an image or visibility data set:
viewerconnection : Scripting connection to the CASA Viewer
vishead     : List, summary, get, and put metadata in a measurement set
visstat     : Displays statistical information from a measurement set, or from a measurement set selection
widefield   : Wide-field imaging and deconvolution with selected algorithm

```

Typing `startup` will provide the startup page displayed when entering CASA. For example,

```

CASA <7>: startup
-----> startup()

```

```

-----
Available tasks:

```

accum	flagmanager	listobs	sdscale
applycal	fluxscale	listvis	sdsim
bandpass	ft	makemask	sdsmooth
blcal	gaincal	plotants	sdstat
browsetable	gencal	plotcal	sdtipimaging
calstat	hanningsmooth	plotmsl	setjy
clean	imcontsub	plotxy	smoothcal
clearcal	imfit	sdaverage	specfit
clearplot	imhead	sdbaseline	split
clearstat	immoments	sdcal	tget
concat	importfits	sdcoadd	tput
deconvolve	importuvfits	sdfit	uvcontsub
exportfits	importvla	sdflag	uvcontsub2
exportuvfits	imregrid	sdimaging	uvmodelfit

feather	imsmooth	sdimprocess	viewer
filecatalog	imstat	sdlist	vishead
find	imval	sdmath	visstat
fixvis	listcal	sdplot	
flagdata	listhistory	sdsave	

sd* tasks are available after `asap_init()` is run

Additional tasks available for ALMA commissioning use
(still alpha code as of Beta release):

simdata	importasdm
---------	------------

Available tools:

cb (calibrator)	cp (cal plot)	fg (flagger)
ia (image analysis)	im (imager)	me (measures)
mp (MS plot)	ms (MS)	qa (quanta)
sm (simulation)	tb (table)	tp (table plot)
vp (voltage patterns)	pm (plotms)	at (atmosphere)

pl (pylab functions)

sd (ASAP functions - run `asap_init()` to import into CASA)

casalogger - Call up the casalogger (if it goes away)

Help:

help taskname	- Full help for task
help par.parametername	- Full help for parameter name
tasklist	- Task list organized by category
taskhelp	- One line summary of available tasks
toolhelp	- One line summary of available tools
startup	- The start up screen

1.3.2 Running Tasks and Tools

Tools are functions linked to the Python interface which must be called by name with arguments. Tasks have higher-level capabilities than tools. Tasks require input parameters which may be specified when you call the task as a function, or be set as parameters in the interface. A task, like a tool, is a function under Python and may be written in Python, C, or C++ (the CASA toolkit is made up of C++ functions).

There are two distinct ways to run tasks. You can either set the global CASA parameters relevant

to the task and tell the task to “go”, or you can call the task as a function with one or more arguments specified. These two invocation methods differ in whether the global parameter values are used or not.

For example,

```
default('plotxy')
vis='ngc5921.ms'
xaxis='channel'
yaxis='amp'
datacolumn='data'
go
```

will execute `plotxy` with the set values for the parameters (see § 1.3.5). Instead of using `go` command (§ 1.3.5.3) to invoke the task, you can also call the task with no arguments, e.g.

```
default('plotxy')
vis='ngc5921.ms'
xaxis='channel'
yaxis='amp'
datacolumn='data'
plotxy()
```

which will also use the global parameter values.

Second, one may call tasks and tools by name with parameters set on the same line. Parameters may be set either as explicit `<parameter>=<value>` arguments, or as a series of comma delimited `<value>`s in the correct order for that task or tool. Note that missing parameters will *use the default values for that task*. For example, the following are equivalent:

```
# Specify parameter names for each keyword input:
plotxy(vis='ngc5921.ms',xaxis='channel',yaxis='amp',datacolumn='data')
# when specifying the parameter name, order doesn't matter, e.g.:
plotxy(xaxis='channel',vis='ngc5921.ms',datacolumn='data',yaxis='amp')
# use parameter order for invoking tasks
plotxy('ngc5921.ms','channel','amp','data')
```

This non-use of globals when calling as a function is so that robust scripts can be written. One need only cut-and-paste the calls and need not worry about the state of the global variables or what has been run previously. It is also more like the standard behavior of function calls in Python and other languages.

Tools can only be called in this second manner by name, with arguments (§ 1.3.6). Tools never use the global parameters and the related mechanisms of `inp` and `go`.

1.3.2.1 Aborting Synchronous Tasks

If you are running CASA tasks synchronously, then you can usually use `CNTL-C` to abort execution of the task. If this does not work, try `CNTL-Z` followed by a `kill`. See § 1.2.5 for more on these methods to abort CASA execution.

You may have to quit and restart CASA after an abort, as the internal state can get mixed up.

1.3.3 Getting Return Values

Some tasks and tools return a record (usually a Python dictionary) to the interface. For example, the `imstat` task (§ 6.7) returns a dictionary with the image statistics in it. To catch these return values into a Python variable, you **MUST** assign that variable to the task call, e.g.

```
xstat = imstat('ngc5921.clean.image')
```

or

```
default('imstat')
imagenname = 'ngc5921.clean.image'
xstat = imstat()
```

Note that tools that return values work in the same way (§ 1.3.6).

You can print or use the return value in Python for controlling scripts. For example,

```
CASA <1>: xstat = imstat('ngc5921.clean.image')
CASA <2>: xstat
Out [2]:
{'blc': array([0, 0, 0, 0]),
 'blcf': '15:24:08.404, +04.31.59.181, I, 1.41281e+09Hz',
 'flux': array([ 4.15292207]),
 'max': array([ 0.05240594]),
 'maxpos': array([134, 134,  0,  38]),
 'maxposf': '15:21:53.976, +05.05.29.998, I, 1.41374e+09Hz',
 'mean': array([ 1.62978083e-05]),
 'medabsdevmed': array([ 0.00127287]),
 'median': array([ -1.10467618e-05]),
 'min': array([-0.0105249]),
 'minpos': array([160,  1,  0,  30]),
 'minposf': '15:21:27.899, +04.32.14.923, I, 1.41354e+09Hz',
 'npts': array([ 3014656.]),
 'quartile': array([ 0.00254587]),
 'rms': array([ 0.00201818]),
 'sigma': array([ 0.00201811]),
 'sum': array([ 49.1322855]),
 'sumsq': array([ 12.27880404]),
 'trc': array([255, 255,  0,  45]),
 'trcf': '15:19:52.390, +05.35.44.246, I, 1.41391e+09Hz'}
CASA <3>: myrms = xstat['rms'][0]
CASA <4>: print 10.0*myrms
0.0201817648485
```

If you do not catch the return variable, it will be lost

```
imstat('ngc5921.clean.image')
```

or

```
default('imstat')
imagenname = 'ngc5921.clean.image'
imstat()
```

and spewed to terminal. Note that `go` will trap and lose the return value, e.g.

```
default('imstat')
imagenname = 'ngc5921.clean.image'
go
```

will not dump the return to the terminal either.

NOTE: You cannot currently catch a return value from a task run asynchronously (§ 1.3.4).

1.3.4 Running Tasks Asynchronously

By default, most tasks run synchronously in the foreground. Many tasks, particularly those that can take a long time to execute, have the `async` parameter. This allows the user to send the task to the background for execution.

ALERT: A few tasks, such as the `exportuvfits` and `exportfits` tasks, have `async=True` by default. This is a workaround for a known problem where they can trample on other tasks and tools if they use the default global tools underneath.

1.3.4.1 Monitoring Asynchronous Tasks

There is a “taskmanager” tool `tm` that allows the user to retrieve the status of, and to abort the execution of, tasks running with `async=True` in the background. There are two methods of interest for the user, `tm.retrieve` and `tm.abort`.

If you run a task with `async=True` then several things will happen. First of all, the task returns a “handle” that is a number used to identify the process. This is printed to the screen, e.g.

```
CASA <5>: inp()
# mosaic :: Calculate a multi-field deconvolved image with selected clean algorithm:
...
async          =          True          # if True run in the background, prompt is freed

CASA <6>: mosaic()
Connecting to controller: ('127.0.0.1', 60775)
Out[6]: 0
```

ALERT:

You should not use the `go` command to run a task asynchronously, as the “handle” will be swallowed by the Python task wrapper and you will not be able to access it with `tm`. This is also true if you run in a Python script.

where the output value 0 is the handle id.

You can also catch the return value in a variable, e.g.

```
CASA <7>: handle = mosaic()
...
CASA <8>: print handle
1
```

You should also see the usual messages from the task in the `logger`, with some extra lines of information

```
#####
### Begin Task: mosaic ###
Tue Oct 2 17:58:16 2007    NORMAL ::mosaic:
""
"Use: "
tm.abort(return_value)    # to abort the asynchronous task
tm.retrieve(return_value) # to retrieve the status
""
... usual messages here ...

### End Task: mosaic ###
#####
""
```

for the example above.

To show the current state of an asynchronous task, use the `tm.retrieve` method using the handle id as the argument. For example,

```
CASA <9>: tm.retrieve(handle)
Out[9]: {'result': None, 'state': 'pending'}
```

or

```
CASA <10>: tm.retrieve(1)
Out[10]: {'result': None, 'state': 'pending'}
```

which means its still running. You should be seeing output in the `logger` also while the task is running.

When a task is finished, you will see:

```
CASA <11>: tm.retrieve(1)
Out[11]: {'result': None, 'state': 'done'}
```

which indicates completion.

1.3.4.2 Aborting Asynchronous Tasks

To abort a task while it is running in the background, use the `tm.abort` method, again with the task handle id as the argument. For example,

```
CASA <12>: handle = mosaic()
...
CASA <13>: tm.abort(handle)
```

will abort the task if it is running.

If this does not work, try `CNTL-Z` followed by a `kill -9 <PID>` for the appropriate process ID. See § 1.2.5 for more on these methods to abort CASA execution.

ALERT: Currently, this is only available with the `tm` tool. We are working on a `taskmanager` task.

1.3.5 Setting Parameters and Invoking Tasks

One can set parameters for tasks (but not for tools) by performing the assignment within the CASA shell and then inspecting them using the `inp` command:

```
CASA <30>: default(bandpass)
CASA <31>: vis = 'ngc5921.demo.ms'
CASA <32>: caltable = 'ngc5921.demo.bcal'
CASA <33>: field = '0'
CASA <34>: refant = '15'
CASA <35>: inp('bandpass')
# bandpass :: Calculates a bandpass calibration solution
vis          = 'ngc5921.demo.ms' # Nome of input visibility file
caltable     = 'ngc5921.demo.bcal' # Name of output gain calibration table
field        = '0'                # Select field using field id(s) or field name(s)
spw          = ''                 # Select spectral window/channels
selectdata   = False             # Other data selection parameters
solint       = 'inf'             # Solution interval
combine      = 'scan'           # Data axes which to combine for solve (scan, spw, field)
refant       = '15'             # Reference antenna name
minblperant  = 4                 # Minimum baselines _per antenna_ required for solve
solnorm      = False            # Normalize average solution amplitudes to 1.0 (G, T only)
bandtype     = 'B'              # Type of bandpass solution (B or BPOLY)
  fillgaps   = 0                 # Fill flagged solution channels by interpolation
append       = False            # Append solutions to the (existing) table
gaintable    = ''               # Gain calibration table(s) to apply on the fly
gainfield    = ''               # Select a subset of calibrators from gaintable(s)
interp       = ''               # Interpolation mode (in time) to use for each gaintable
spwmap       = []               # Spectral windows combinations to form for gaintables(s)
gaincurve    = False            # Apply internal VLA antenna gain curve correction
opacity      = 0.0              # Opacity correction to apply (nepers)
```

Inside the Toolkit:

In the current version of CASA, you cannot use the task parameter setting features, such as the `inp`, `default`, or `go` commands, for the tools.


```
parang      =      False      # Apply parallactic angle correction
async      =      False      #
```

See § 1.3.5.4 below for more details on the use of the `inputs` command.

All task parameters have **global** scope within CASA: the parameter values are common to all tasks and also at the CASA command line. This allows the convenience of not changing parameters that are shared between tasks but does require care when chaining together sequences of task invocations (to ensure proper values are provided).

If you want to reset the input keywords for a single task, use the `default` command (§ 1.3.5.2). For example, to set the defaults for the `bandpass` task, type:

```
CASA <30>: default('bandpass')
```

as we did above.

To inspect a single parameter value just type it at the command line. Continuing the above example:

```
CASA <36>: combine
          Out[14]: 'scan'
```

CASA parameters are just Python variables.

Parameters for a given task can be saved by using the `saveinputs` command (see § 1.3.5.5) and restored using the `execfile '<filename>'` command. Note that if the task is successfully executed, then a `<taskname>.last` file is created in the working directory containing the parameter values (see § 1.3.5.8).

We now describe the individual CASA task parameter interface commands and features in more detail.

1.3.5.1 The scope of parameters in CASA

All task parameters have **global** scope within CASA: the parameter values are common to all tasks and also at the CASA command line. This allows the convenience of not changing parameters that are shared between tasks but does require care when chaining together sequences of task invocations (to ensure proper values are provided). Tasks **DO NOT** change the values of the global parameters, nor does the invocation of tasks using the functional call with arguments change the globals.

This does mean that unless you do an explicit `default` of the task (§ 1.3.5.2), previously set values may be unexpectedly used if you do not inspect the `inp` carefully. For example, good practice is:

Advanced Tip

By default, the scope of CASA parameters is global, as stated here. However, if you call a task as a function with one or more arguments specified, e.g. `task(arg1=val1, ...)`, then non-specified parameters will be defaulted and no globals used. This makes scripting more robust. Tasks **DO NOT** change the value of globals.

```

default('imhead')
imagenname = 'ngc5921.demo.cleanimg.image'
mode = 'list'
imhead()

```

If you supply the task call with arguments, then these will be used for the values of those parameters (see above). However, if some but not all arguments are supplied, then those parameters not given as arguments will default and NOT use the current global values. Thus,

```
imhead('ngc5921.demo.cleanimg.image',mode='list')
```

will reproduce the above.

1.3.5.2 The default Command

Each task has a special set of default parameters defined for its parameters. You can use the `default` command to reset the parameters for a specified task (or the current task as defined by the `taskname` variable) to their default.

Important Note: The `default` command resets the values of the task parameters to a set of “defaults” as specified in the task code. Some defaults are blank strings `''` or empty lists `[]`, others are specific numerical values, strings, or lists. It is important to understand that just setting a string parameter to an empty string `''` is not setting it to its default! Some parameters do not have a blank as an allowed value. See the `help` for a particular task to find out its default. If `''` is the default or an allowed value, it will say so explicitly.

For example, suppose we have been running CASA on a particular dataset, e.g.

```

CASA <40>: inp clean
-----> inp('clean')
# clean :: Deconvolve an image with selected algorithm
vis                = 'ngc5921.demo.src.split.ms.contsub' # name of input visibility file
imagenname         = 'ngc5921.demo.cleanimg' # Pre-name of output images
field              = '0' # Field Name
spw                = '' # Spectral windows:channels: '' is all
selectdata         = False # Other data selection parameters
mode               = 'channel' # Type of selection (mfs, channel, velocity, frequency)
    nchan          = 46 # Number of channels (planes) in output image
    start          = 5 # first input channel to use
    width          = 1 # Number of input channels to average
    interpolation   = 'nearest' # Spectral interpolation (nearest, linear, cubic)
niter              = 6000 # Maximum number of iterations
...

```

and now we wish to switch to a different one. We can reset the parameter values using `default`:

```

CASA <41>: default
-----> default()

CASA <42>: inp
-----> inp()
# clean :: Deconvolve an image with selected algorithm
vis          =      ''          # name of input visibility file
imagename    =      ''          # Pre-name of output images
field        =      ''          # Field Name
spw          =      ''          # Spectral windows:channels: '' is all
selectdata   =      False       # Other data selection parameters
mode         =      'mfs'       # Type of selection (mfs, channel, velocity, frequency)
niter        =      500         # Maximum number of iterations
...

```

It is good practice to use `default` before running a task if you are unsure what state the CASA global variables are in.

ALERT: You currently can only reset ALL of the parameters for a given task to their defaults. In an upcoming update we will allow the `default` command to take a second argument with a specific parameter to default its value.

1.3.5.3 The go Command

You can execute a task using the `go` command, either explicitly

```

CASA <44>: go listobs
-----> go(listobs)
Executing: listobs()
...

```

or implicitly if `taskname` is defined (e.g. by previous use of `default` or `inp`)

```

CASA <45>: taskname = 'clean'
CASA <46>: go
-----> go()
Executing: clean()
...

```

You can also execute a task simply by typing the `taskname`.

```

CASA <46>: clean
-----> clean()
Executing: clean()
...

```

The `go` command can also be used to launch a different task without changing the current `taskname`, without disrupting the `inp` process on the current task you are working on. For example

```

default 'gaincal' # set current task to gaincal and default
vis = 'n5921.ms' # set the working ms
... # set some more parameters
go listobs # launch listobs w/o changing current task
inp # see the inputs for gaincal (not listobs!)

```

ALERT: Doing `go listobs(vis='foo.ms')` will currently change the `taskname`, and will change `vis`, which might not be what is desired.

1.3.5.4 The `inp` Command

You can set the values for the parameters for tasks (but currently not for tools) by performing the assignment within the CASA shell and then inspecting them using the `inp` command. This command can be invoked in any of three ways: via function call `inp('<taskname>')` or `inp(<taskname>)`, without parentheses `inp '<taskname>'` or `inp <taskname>`, or using the current `taskname` variable setting with `inp`. For example,

```

CASA <1>: inp('clean')
...
CASA <2>: inp 'clean'
-----> inp('clean')
...
CASA <3>: inp(clean)
...
CASA <4>: inp clean
-----> inp(clean)
...
CASA <5>: taskname = 'clean'
CASA <6>: inp
-----> inp()

```

all do the same thing.

When you invoke the task inputs via `inp`, you see a list of the parameters, their current values, and a short description of what that parameters does. For example, starting from the default values,

```

CASA <18>: inp('clean')
# clean :: Deconvolve an image with selected algorithm
vis           =           ''           # name of input visibility file
imagenam     =           ''           # Pre-name of output images
field        =           ''           # Field Name
spw          =           ''           # Spectral windows:channels: '' is all
selectdata   =      False           # Other data selection parameters
mode         =      'mfs'           # Type of selection (mfs, channel, velocity, frequency)
niter        =           500          # Maximum number of iterations
gain         =           0.1          # Loop gain for cleaning
threshold    =      '0.0mJy'        # Flux level to stop cleaning. Must include units
psfmode      =      'clark'          # method of PSF calculation to use during minor cycles
imagermode   =           ''           # Use csclean or mosaic. If '', use psfmode

```

```

multiscale      =      []      # multi-scale deconvolution scales (pixels)
interactive     =      False   # use interactive clean (with GUI viewer)
mask           =      []      # cleanbox(es), mask image(s), and/or region(s)
imsize        = [256, 256]    # x and y image size in pixels
cell          = ['1.0arcsec', '1.0arcsec'] # x and y cell size. default unit arcsec
phasecenter    =      ''      # Image phase center: position or field index
restfreq      =      ''      # rest frequency to assign to image (see help)
stokes        =      'I'      # Stokes params to image (eg I,IV, QU,IQUV)
weighting     = 'natural'    # Weighting of uv (natural, uniform, briggs, ...)
uvtaper       =      False   # Apply additional uv tapering of visibilities.
modelimage    =      ''      # Name of model image(s) to initialize cleaning
restoringbeam =      ['']    # Output Gaussian restoring beam for CLEAN image
pbcor         =      False   # Output primary beam-corrected image
minpb        =      0.1      # Minimum PB level to use
async        =      False   # If true the taskname must be started using clean(...)

```

Figure 1.1 shows how this will look to you on your terminal. Note that some parameters are in boldface with a gray background. This means that some values for this parameter will cause it to *expand*, revealing new *sub-parameters* to be set.

```

xterm <5>
CASA <2>: default('clean')
CASA <3>: inp('clean')
# clean :: Deconvolve an image with selected algorithm
vis          =      ''      # name of input visibility file
imagename    =      ''      # Pre-name of output images
field       =      ''      # Field Name
spw         =      ''      # Spectral windows:channels: '' is all
selectdata  =      False   # Other data selection parameters
mode        =      'mfs'   # Type of selection (mfs, channel, velocity, frequency)
niter       =      500     # Maximum number of iterations
gain        =      0.1     # Loop gain for cleaning
threshold   = '0.0mJy'    # Flux level to stop cleaning. Must include units
psfmode     =      'clark' # method of PSF calculation to use during minor cycles
imagermode  =      ''      # Use csclean or mosaic. If '', use psfmode
multiscale =      []      # set deconvolution scales (pixels), default: multiscale=[] (standard CLEAN)
interactive =      False # use interactive clean (with GUI viewer)
mask        =      []      # cleanbox(es), mask image(s), and/or region(s) used in cleaning
imsize     = [256, 256]    # x and y image size in pixels, symmetric for single value
cell       = ['1.0arcsec', '1.0arcsec'] # x and y cell size. default unit arcsec
phasecenter =      ''      # Image phase center: position or field index
restfreq   =      ''      # rest frequency to assign to image (see help)
stokes     =      'I'      # Stokes params to image (eg I,IV, QU,IQUV)
weighting  = 'natural'    # Weighting of uv (natural, uniform, briggs, ...)
uvtaper    =      False   # Apply additional uv tapering of visibilities.
modelimage =      ''      # Name of model image(s) to initialize cleaning
restoringbeam =      [''] # Output Gaussian restoring beam for CLEAN image
pbcor      =      False   # Output primary beam-corrected image
minpb     =      0.1      # Minimum PB level to use
async     =      False   # If true the taskname must be started using clean(...)
CASA <4>: █

```

Figure 1.1: Screen shot of the default CASA inputs for task `clean`.

CASA uses color and font to indicate different properties of parameters and their values:

Parameter and Values in CASA inp

	Text Font	Text Color	Highlight	Indentation	Meaning
Parameters:					
	plain	black	none	none	standard parameter
	bold	black	grey	none	expandable parameter
	plain	green	none	yes	sub-parameter
Values:					
	plain	black	none	none	default value
	plain	blue	none	none	non-default value
	plain	red	none	none	invalid value

Figure 1.2 shows what happens when you set some of the `clean` parameters to non-default values. Some have opened up sub-parameters, which can now be seen and set. Figure 1.3 shows what happens when you set a parameter, in this case `vis` and `mode`, to an invalid value. Its value now appears in red. Reasons for invalidation include incorrect type, an invalid menu choice, or a filename that does not exist. For example, since `vis` expects a filename, it will be invalidated (red) if it is set to a non-string value, or a string that is not the name of a file that can be found. The `mode='happy'` is invalid because its not a supported choice ('mfs', 'channel', 'velocity', or 'frequency').

```

xterm <5>
CASA <4>: tget('clean')
Restored parameters from file clean.last

CASA <5>: inp('clean')
# clean :: Deconvolve an image with selected algorithm
vis          = 'ngc5921.demo.src.split.ms.contsub' # name of input visibility file
imagenam    = 'ngc5921.demo.cleaning' # Pre-name of output images
field       = '0' # Field Name
spw         = '' # Spectral windows:channels: '' is all
selectdata  = False # Other data selection parameters
mode       = 'channel' # Type of selection (mfs, channel, velocity, frequency)
  nchan     = 46 # Number of channels (planes) in output image
  start     = 5 # first input channel to use
  width     = 1 # Number of input channels to average
  interpolation = 'nearest' # Type of spectral interpolation of visibilities (nearest, linear, cubic)

niter       = 6000 # Maximum number of iterations
gain        = 0.1 # Loop gain for cleaning
threshold   = 8.0 # Flux level to stop cleaning. Must include units
psfmode     = 'hogbom' # method of PSF calculation to use during minor cycles
imagermode = '' # Use csclean or mosaic. If '', use psfmode
multiscale = [] # set deconvolution scales (pixels), default: multiscale=[] (standard CLEAN)
interactive = False # use interactive clean (with GUI viewer)
mask        = [108, 108, 148, 148] # cleanbox(es), mask image(s), and/or region(s) used in cleaning
imsz       = [256, 256] # x and y image size in pixels, symmetric for single value
cell        = [15.0, 15.0] # x and y cell size. default unit arcsec
phasecenter = '' # Image phase center: position or field index
restfreq    = '' # rest frequency to assign to image (see help)
stokes      = 'I' # Stokes params to image (eg I,IV,QU,IQUV)
weighting  = 'briggs' # Weighting of uv (natural, uniform, briggs, ...)
  robust    = 0.5 # Briggs robustness parameter
  npixels   = 0 # number of pixels to determine uv-cell size 0=> field of view

uvtaper   = False # Apply additional uv tapering of visibilities.
modelimage = '' # Name of model image(s) to initialize cleaning.
restoringbeam = [''] # Output Gaussian restoring beam for CLEAN image
pbcor      = False # Output primary beam-corrected image
minpb     = 0.1 # Minimum PB level to use
async     = False # If true the taskname must be started using clean(...)

CASA <6>:

```

Figure 1.2: The `clean` inputs after setting values away from their defaults (blue text). Note that some of the boldface ones have opened up new dependent sub-parameters (indented and green).

```

xterm <5>
CASA <6>: psfmode = 'hogwarts'
CASA <7>: inp('clean')
# clean :: Deconvolve an image with selected algorithm
vis          = 'ngc5921.demo.src.split.ms.contsub' # name of input visibility file
imagename    = 'ngc5921.demo.cleaning' # Pre-name of output images
field        = '0' # Field Name
spw          = '' # Spectral windows:channels: '' is all
selectdata   = False # Other data selection parameters
mode         = 'channel' # Type of selection (mfs, channel, velocity, frequency)
  nchan      = 46 # Number of channels (planes) in output image
  start      = 5 # first input channel to use
  width      = 1 # Number of input channels to average
  interpolation = 'nearest' # Type of spectral interpolation of visibilities (nearest, linear, cubic)

niter        = 6000 # Maximum number of iterations
gain         = 0.1 # Loop gain for cleaning
threshold    = 8.0 # Flux level to stop cleaning. Must include units
psfmode      = 'hogwarts' # method of PSF calculation to use during minor cycles
imagermode   = '' # Use csclean or mosaic. If '', use psfmode
multiscale   = [] # set deconvolution scales (pixels), default: multiscale=[] (standard CLEAN)
interactive   = False # use interactive clean (with GUI viewer)
mask         = [108, 108, 148, 148] # cleanbox(es), mask image(s), and/or region(s) used in cleaning
imsz         = [256, 256] # x and y image size in pixels, symmetric for single value
cell         = [15.0, 15.0] # x and y cell size. default unit arcsec
phasecenter  = '' # Image phase center: position or field index
restfreq     = '' # rest frequency to assign to image (see help)
stokes       = 'I' # Stokes params to image (eg I,IV,QU,IQUV)
weighting    = 'briggs' # Weighting of uv (natural, uniform, briggs, ...)
  robust     = 0.5 # Briggs robustness parameter
  npixels    = 0 # number of pixels to determine uv-cell size 0=> field of view

uvtaper      = False # Apply additional uv tapering of visibilities.
modelimage   = '' # Name of model image(s) to initialize cleaning.
restoringbeam = [] # Output Gaussian restoring beam for CLEAN image
pbcor        = False # Output primary beam-corrected image
minpb        = 0.1 # Minimum PB level to use
async        = False # If true the taskname must be started using clean(...)

CASA <8>:

```

Figure 1.3: The `clean` inputs where one parameter has been set to an invalid value. This is drawn in red to draw attention to the problem. This hapless user probably confused the `'hogbom'` clean algorithm with Harry Potter.

1.3.5.5 The `saveinputs` Command

The `saveinputs` command will save the current values of a given task parameters to a Python (plain ascii) file. It can take up to two arguments, e.g.

```
saveinputs(taskname, outfile)
```

The first is the usual `taskname` parameter. The second is the name for the output Python file. If there is no second argument, for example,

```
saveinputs('clean')
```

a file with name `<taskname>.saved` (in this case `'clean.saved'` will be created or overwritten if extant. If invoked with no arguments, e.g.

```
saveinputs
```

it will use the current values of the `taskname` variable (as set using `inp <taskname>` or default `<taskname>`). You can also use the `taskname` global parameter explicitly,

```
saveinputs(taskname, taskname+'_1.save')
```

For example, starting from default values

```
CASA <1>: default('listobs')
CASA <2>: vis='ngc5921.demo.ms'
CASA <3>: saveinputs
CASA <4>: !more 'listobs.saved'
IPython system call: more 'listobs.saved'
taskname          = "listobs"
vis               = "ngc5921.demo.ms"
verbose          = True
#listobs(vis="ngc5921.ms",verbose=False)
```

To read these back in, use the Python `execfile` command. For example,

```
CASA <5>: execfile('listobs.saved')
```

and we are back.

An example save to a custom named file:

```
CASA <6>: saveinputs('listobs','ngc5921_listobs.par')
```

You can also use the CASA `tget` command (see § 1.3.5.6 below) instead of the Python `execfile` to restore your inputs.

1.3.5.6 The `tget` Command

The `tget` command will recover saved values of the inputs of tasks. This is a convenient alternative to using the Python `execfile` command (see above).

Typing `tget` without a taskname will recover the saved values of the inputs for the current task as given in the current value of the `taskname` parameter.

Adding a task name, e.g. `tget <taskname>` will recover values for the specified task. This is done by searching for 1) a `<taskname>.last` file (see § 1.3.5.8 below), then for 2) a `<taskname>.saved` file (see § 1.3.5.5 above), and then executing the Python in these files.

For example,

```
default('gaincal') # set current task to gaincal and default
tget              # read saved inputs from gaincal.last (or gaincal.saved)
inp              # see these inputs!
tget bandpass    # now get from bandpass.last (or bandpass.saved)
inp              # task is now bandpass, with recovered inputs
```


1.3.5.7 The tput Command

The `tput` command will save the current parameter values of a task to its `<taskname>.last` file. This is a shorthand to `saveinputs` and is a counterpart to `tget`.

Typing `tput` without a taskname will save the values of the inputs for the current task as given in the current value of the `taskname` parameter.

Adding a task name, e.g. `tget <taskname>` will save the values for the specified task.

For example,

```
default('gaincal') # set current task to gaincal and default
tget               # read saved inputs from gaincal.last (or gaincal.saved)
inp               # see these inputs!
vis = 'new.ms'    # change the vis parameter
tput              # save back to the gaincal.last file for later use
```

1.3.5.8 The .last file

Whenever you successfully execute a CASA task, a Python script file called `<taskname>.last` will be written (or over-written) into the current working directory. For example, if you ran the `listobs` task as detailed above, then

```
CASA <14>: vis = 'ngc5921.ms'
CASA <15>: verbose = True
CASA <16>: listobs()
CASA <17>: !more 'listobs.last'
IPython system call: more listobs.last
taskname      = "listobs"
vis           = "ngc5921.ms"
verbose       = True
#listobs(vis="ngc5921.ms",verbose=False)
```

You can restore the parameter values from the save file using

```
CASA <18>: execfile('listobs.last')
```

or

```
CASA <19>: run listobs.last
```

Note that the `.last` file is generally not created until the task actually finished (successfully), so it is often best to manually create a save file beforehand using the `saveinputs` command if you are running a critical task that you strongly desire to have the inputs saved for.

1.3.6 Tools in CASA

The *CASA toolkit* is the foundation of the functionality in the package, and consists of a suite of functions that are callable from Python. The tools are used by the tasks, and can be used by advanced users to perform operations that are not available through the tasks.

It is beyond the scope of this reference to describe the toolkit in detail. Occasionally, examples will be given that utilize the tools (e.g. § 6.12). In short, tools are always called as functions, with any parameters that are not to be defaulted given as arguments. For example:

```
ia.open('ngc5921.chan21.clean.cleanbox.mask')
ia.calcmask('"ngc5921.chan21.clean.cleanbox.mask">0.5', 'mymask')
ia.summary()
ia.close()
```

uses the `image` tool (`ia`) to turn a clean mask image into an image mask. Tools never use the CASA global parameters.

To find what tools are available, use the `toolhelp` command:

```
CASA <8>: toolhelp
-----> toolhelp()
```

Available tools:

```
at : Juan Pardo ATM library
cb : Calibration utilities
cp : Cal solution plotting utilities
fg : Flagging/Flag management utilities
ia : Image analysis utilities
im : Imaging utilities
me : Measures utilities
ms : MeasurementSet (MS) utilities
mp : MS plotting (data (amp/phase) versus other quantities)
pm : PlotMS utilities
tb : Table utilities (selection, extraction, etc)
tp : Table plotting utilities
qa : Quanta utilities
sm : Simulation utilities
vp : Voltage pattern/primary beam utilities
---
pl : pylab functions (e.g., pl.title, etc)
sd : (after running asap_init()) Single dish utilities
---
```

You can find much more information about the toolkit in the *CASA User Reference Manual*:

<http://casa.nrao.edu/docs/casaref/CasaRef.html>

1.4 Getting the most out of CASA

There are some other general things you should know about using CASA in order to make things go smoothly during your data reduction.

1.4.1 Your command line history

Your command line history is automatically maintained and stored as `ipython.log` in your local directory `.` This file can be edited and re-executed as appropriate using the `execfile '<filename>'` feature.

You can also use the “up-arrow” and “down-arrow” keys for command line recall in the `casapy` interface. If you start typing text, and then use “up-arrow”, you will navigate back through commands matching what you typed.

1.4.2 Logging your session

The output from CASA commands is sent to the file `casapy.log`, also in your local directory. Whenever you start up `casapy`, the previous `casapy.log` is renamed (based on the date and time) and a new log file is started.

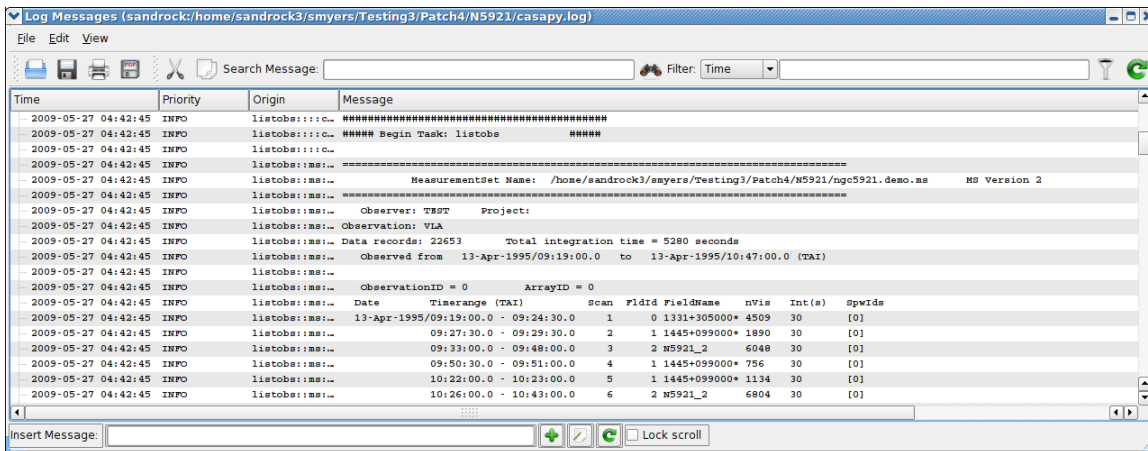


Figure 1.4: The CASA Logger GUI window under Linux. Note that under MacOSX a stripped down logger will instead appear as a Console.

The output contained in `casapy.log` is also displayed in a separate window using the `casalogger`. Generally, the logger window will be brought up when `casapy` is started. If you do not want the logger GUI to appear, then start `casapy` using the `--nologger` option,

```
casapy --nologger
```

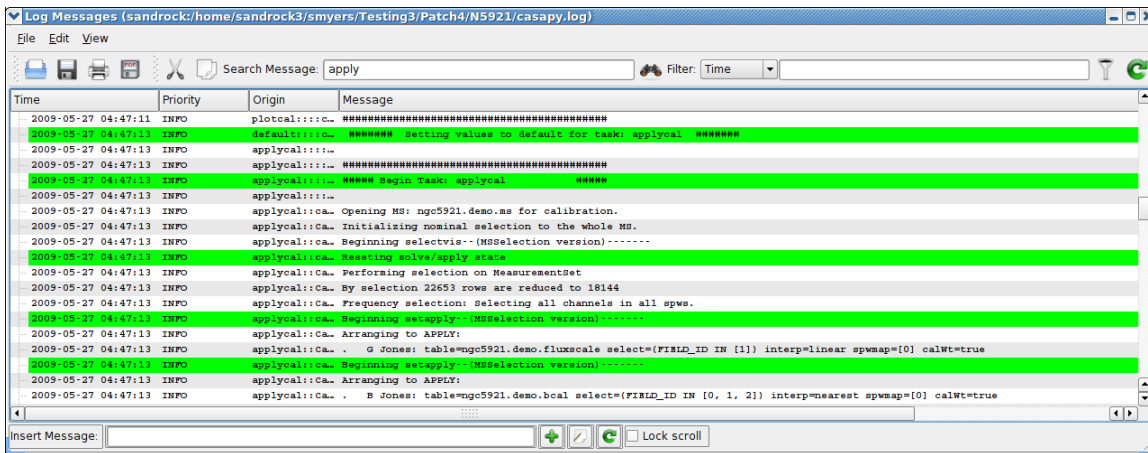


Figure 1.5: Using the Search facility in the `casalogger`. Here we have specified the string 'plotted' and it has highlighted all instances in green.

which will run CASA in the terminal window. See § 1.4.2.1 for more startup options.

ALERT: Due to problems with Qt under MacOSX, we had to replace the GUI `qtcasalogger` with a special stripped down one that uses the Mac Console. This still has the important capabilities such as showing the messages and cut/paste. The following description is for the Linux version and thus should mostly be disregarded on OSX. On the Mac, you treat this as just another console window and use the usual mouse and hot-key actions to do what is needed.

The CASA logger window for Linux is shown in Figure 1.4. The main feature is the display area for the log text, which is divided into columns. The columns are:

- **Time** — the time that the message was generated. Note that this will be in local computer time (usually UT) for casapy generated messages, and may be different for user generated messages;
- **Priority** — the *Priority Level* (see below) of the message;
- **Origin** — where within CASA the message came from. This is in the format `Task::Tool::Method` (one or more of the fields may be missing depending upon the message);
- **Message** — the actual text.

The `casalogger` GUI has a range of features, which include:

- **Search** — search messages by entering text in the Search window and clicking the search icon. The search currently just matches the exact text you type anywhere in the message. See Figure 1.5 for an example.

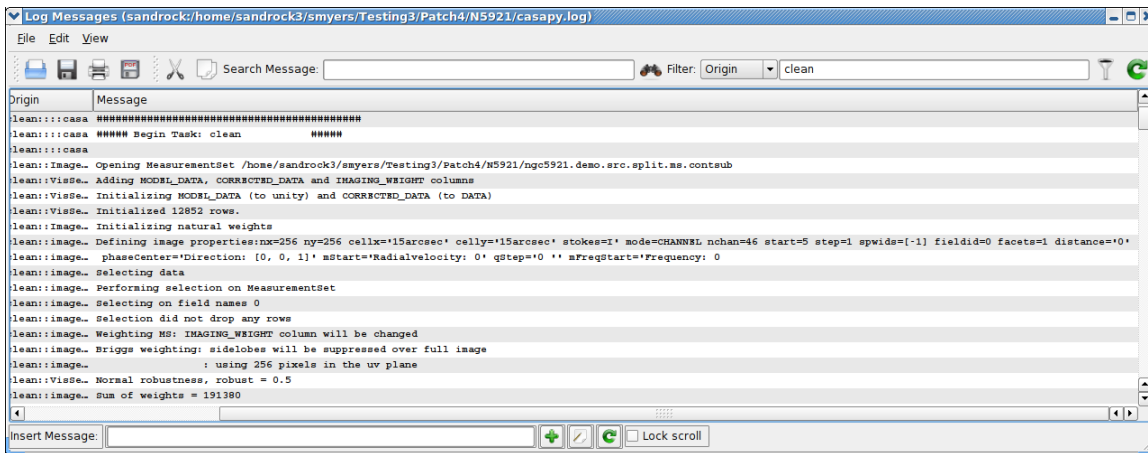


Figure 1.6: Using the `casalogger` Filter facility. The log output can be sorted by Priority, Time, Origin, and Message. In this example we are filtering by Origin using `'clean'`, and it now shows all the log output from the `clean` task.

- **Filter** — a filter to sort by message priority, time, task/tool of origin, and message contents. Enter text in the Filter window and click the filter icon to the right of the window. Use the pull-down at the left of the Filter window to choose what to filter. The matching is for the exact text currently (no regular expressions). See Figure 1.6 for an example.
- **View** — show and hide columns (Time, Priority, Origin, Message) by checking boxes under the **View** menu pull-down. You can also change the font here.
- **Insert Message** — insert additional comments as “notes” in the log. Enter the text into the “Insert Message” box at the bottom of the logger, and click on the Add (+) button, or choose to enter a longer message. The entered message will appear with a priority of “NOTE” with the Origin as your username. See Figure 1.7 for an example. **ALERT:** This message currently will not be inserted into the correct (or user controllable) order into the log.
- **Copy** — left-click on a row, or click-drag a range of rows, or click at the start and shift click at the end to select. Use the Copy button or **Edit** menu Copy to put the selected rows into the clipboard. You can then (usually) paste this where you wish. **ALERT:** this does not work routinely in the current version. You are best off going to the `casapy.log` file if you want to grab text.
- **Open** — There is an Open function in the **File** menu, and an Open button, that will allow you to load old `casalogger` files. **ALERT:** You cannot see the old `casapy.log` files with timestamps (e.g. names like `casapy.log-2009-05-26T22:24:31`) currently in the file browser, but you can type the name in explicitly and hit `<return>` to view them.

Other operations are also possible from the menu or buttons. Mouse “flyover” will reveal the operation of buttons, for example.

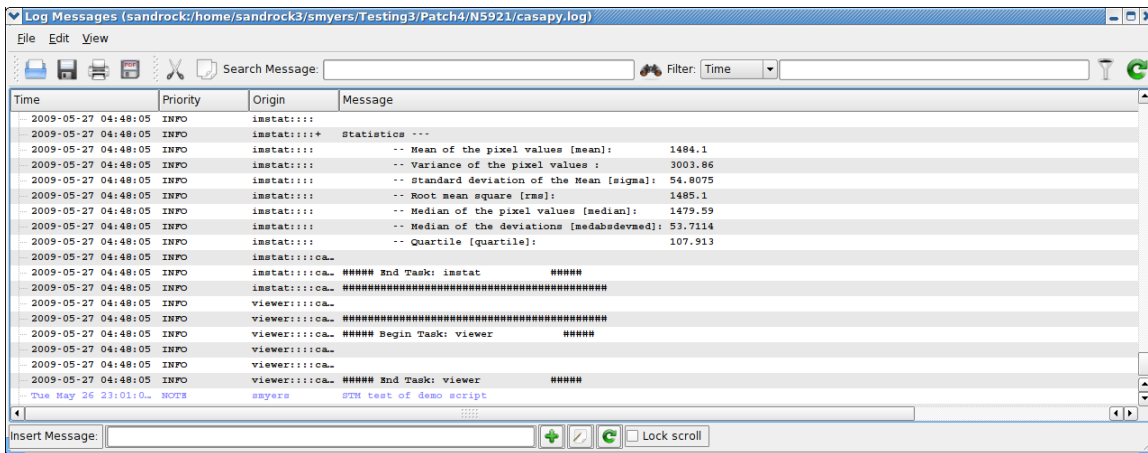


Figure 1.7: CASA Logger - Insert facility: The log output can be augmented by adding notes or comments during the reduction. The file should then be saved to disk to retain these changes.

1.4.2.1 Startup options for the logger

One can specify logger options at the startup of `casapy` on the command line:

```
casapy <logger option>
```

These options are:

```
--log2term           == logging message go to terminal
--nologfile          == no casapy.log logfile is produced
--logfile <filename> == use specified name for logfile instead of casapy.log
--nologger           == do not bring up GUI logger (see above)
--nolog (is deprecated use --nologger)
```

For example, to not bring up a GUI but send the message to your terminal, do

```
casapy --nologger --log2term
```

while

```
casapy --logfile mynewlogfile.log
```

will start `casapy` with logger messages going to the file `mynewlogfile.log`.

1.4.2.2 Setting priority levels in the logger

Logger messages are assigned a *Priority Level* when generated within CASA. The current levels of Priority are:

1. **SEVERE** — errors;
2. **WARN** — warnings;
3. **INFO** — basic information every user should be aware of or has requested;
4. **INFO1** — information possibly helpful to the user;
5. **INFO2** — details the power user might want to see;
6. **INFO3** — even more details;
7. **INFO4** — lowest level of non-debugging information;
8. **DEBUGGING** — most “important” debugging messages;
9. **DEBUG1** — more details;
10. **DEBUG2** — lowest level of debugging messages.

The “debugging” levels are intended for the developers use.

There is a threshold for which these messages are written to the `casapy.log` file and are thus visible in the logger. By default, only messages at level **INFO** and above are logged. The user can change the threshold using the `casalog.filter` method. This takes a single string argument of the `level` for the threshold. The `level` sets the lowest priority that will be generated, and all messages of this level or higher will go into the `casapy.log` file.

Some examples:

```
casalog.filter('INFO')           # the default
casalog.filter('INFO2')         # should satisfy even advanced users
casalog.filter('INFO4')         # all INFOx messages
casalog.filter('DEBUG2')        # all messages including debugging
```

Inside the Toolkit:

The `casalog` tool can be used to control the logging. In particular, the `casalog.filter` method sets the priority threshold. This tool can also be used to change the output log file, and to post messages into the logger.

WARNING: Setting the threshold to **DEBUG2** will put lots of messages in the log!

1.4.3 Where are my data in CASA?

Interferometric data are filled into a so-called Measurement Set (or MS). In its logical structure, the MS looks like a generalized description of data from any interferometric or single dish telescope. Physically, the MS consists of several tables in a directory on disk.

Tables in CASA are actually directories containing files that are the sub-tables. For example, when you create a MS called `AM675.ms`, then the name of the directory where all the tables are stored will be called `AM675.ms/`. See Chapter 2 for more information on Measurement Set and Data Handling in CASA.

The data that you originally get from a telescope can be put in any directory that is convenient to you. Once you "fill" the data into a measurement set that can be accessed by CASA, it is generally best to keep that MS in the same directory where you started CASA so you can get access to it easily (rather than constantly having to specify a full path name).

When you generate calibration solutions or images (again these are in table format), these will also be written to disk. It is a good idea to keep them in the directory in which you started CASA.

1.4.3.1 How do I get rid of my data in CASA?

Note that when you delete a measurement set, calibration table, or image, which are in fact directories, you must delete this and all underlying directories and files. If you are not running `casapy`, this is most simply done by using the file delete method of the operating system you started CASA from. For example, when running CASA on a Linux system, in order to delete the measurement set named `AM675.ms` type:

```
CASA <5>: !rm -r AM675.ms
```

from within CASA. The `!` tells CASA that a system command follows (see § 1.2.7.5), and the `-r` makes sure that all subdirectories are deleted recursively.

It is convenient to prefix all MS, calibration tables, and output files produced in a run with a common string. For example, one might prefix all files from VLA project AM675 with `AM675`, e.g. `AM675.ms`, `AM675.cal`, `AM675.clean`. Then,

```
CASA <6>: !rm -r AM675*
```

will clean up all of these.

In scripts, the `!` escape to the OS will not work. Instead, use the `os.system()` function (Appendix D.6.1) to do the same thing:

```
os.system('rm -r AM675*')
```

If you are within `casapy`, then the CASA system is keeping a cache of tables that you have been using and using the OS to delete them will confuse things. For example, running a script that contains `rm` commands multiple times will often not run or crash the second time as the cache gets confused. The clean way of removing CASA tables (MS, caltables, images) inside `casapy` is to use the `rmtables` task:


```
rmtables('AM675.ms')
```

and this can also be wildcarded

```
rmtables('AM675*')
```

(though you may get warnings if it tries to delete files or directories that fit the name wildcard that are not CASA tables).

ALERT: Some CASA processes lock the file and forget to give it up when they are done (`plotxy` is usually the culprit). You will get `WARNING` messages from `rmtables` and your script will probably crash second time around as the file isn't removed. The safest thing is still to exit `casapy` and start a new session for multiple runs.

1.4.4 What's in my data?

The actual data is in a large `MAIN` table that is organized in such a way that you can access different parts of the data easily. This table contains a number of “rows”, which are effectively a single timestamp for a single spectral window (like an IF from the VLA) and a single baseline (for an interferometer).

There are a number of “columns” in the MS, the most important of which for our purposes is the `DATA` column — this contains the original visibility data from when the MS was created or filled. There are other helpful “scratch” columns which hold useful versions of the data or weights for further processing: the `CORRECTED_DATA` column, which is used to hold calibrated data; the `MODEL_DATA` column, which holds the Fourier inversion of a particular model image; and the `IMAGING_WEIGHT` column which can hold the weights to be used in imaging. The creation and use of the scratch columns is generally done behind the scenes, but you should be aware that they are there (and when they are used). We will occasionally refer to the rows and columns in the MS.

More on the contents of the MS can be found in § 2.1.

1.4.5 Data Selection in CASA

We have tried to make the CASA task interface as uniform as possible. If a given parameter appears in multiple tasks, it should, as far as is possible, mean the same thing and be used in the same way in each. There are groups of parameters that appear in a number of tasks to do the same thing, such as for data selection.

The parameters `field`, `spw`, and `selectdata` (which if `True` expands to a number of sub-parameters) are commonly used in tasks to select data on which to work. These common data selection parameters are described in § 2.3.

1.5 From Loading Data to Images

The subsections below provide a brief overview of the steps you will need to load data into CASA and obtain a final, calibrated image. Each subject is covered in more detail in Chapters 2 through 6.

An end-to-end workflow diagram for CASA data reduction for interferometry data is shown in Figure 1.8. This might help you chart your course through the package. In the following subsections, we will chart a rough course through this process, with the later chapters filling in the individual boxes.

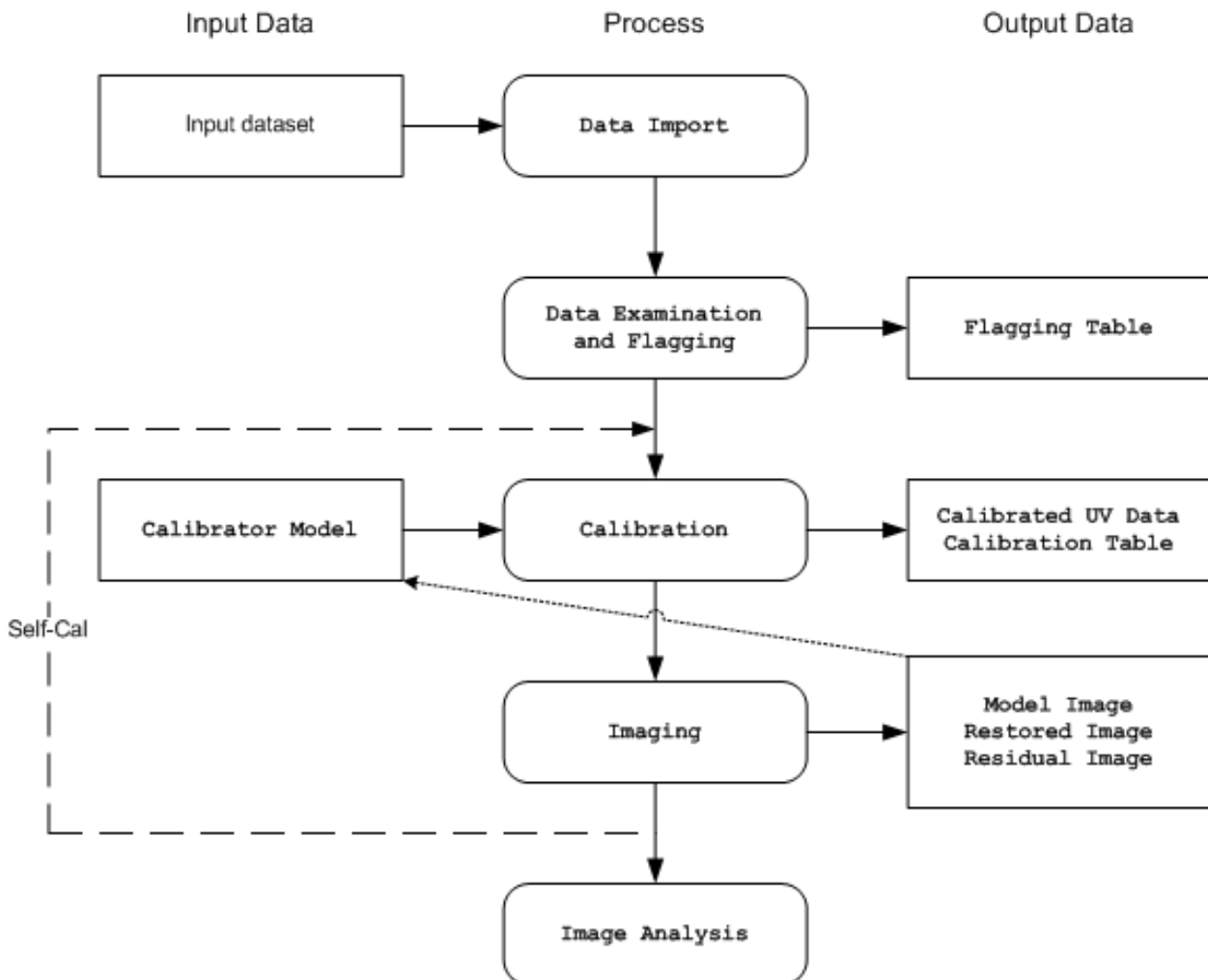


Figure 1.8: Flow chart of the data processing operations that a general user will carry out in an end-to-end CASA reduction session.

Note that single-dish data reduction (for example with the ALMA single-dish system) follows a similar course. This is detailed in Chapter A.

1.5.1 Loading Data into CASA

The key data and image import tasks are:

- `importuvfits` — import visibility data in UVFITS format (§ 2.2.3);
- `importvla` — import data from VLA that is in *export* format (§ 2.2.2);
- `importasdm` — import ALMA or EVLA data in ASDM format (§ 2.2.1);
- `importfits` — import a FITS image into a CASA *image* format table (§ 6.11).

These are used to bring in your interferometer data, to be stored as a CASA Measurement set (MS), and any previously made images or models (to be stored as CASA image tables).

The data import tasks will create a MS with a path and name specified by the `vis` parameter. See § 1.4.3 for more information on MS in CASA. The measurement set is the internal data format used by CASA, and conversion from any other native format is necessary for most of the data reduction tasks.

Once data is imported, there are other operations you can use to manipulate the datasets:

- `concat` — concatenate a second MS into a given MS (§ 2.2.9)

Data import, export, concatenation, and selection detailed in Chapter 2.

1.5.1.1 VLA: Filling data from VLA archive format

VLA data in “archive” format are read into CASA from disk using the `importvla` task (see § 2.2.2). This filler supports the new naming conventions of EVLA antennas when incorporated into the old VLA system.

Note that future data from the EVLA in ASDM format will use a different filler. This will be made available in a later release.

1.5.1.2 Filling data from UVFITS format

For UVFITS format, use the `importuvfits` task. A subset of popular flavors of UVFITS (in particular UVFITS as written by AIPS) is supported by the CASA filler. See § 2.2.3 for details.

1.5.1.3 Loading FITS images

For FITS format images, such as those to be used as calibration models, use the `importfits` task. Most, though not all, types of FITS images written by astronomical software packages can be read in.

See § 6.11 for more information.

1.5.1.4 Concatenation of multiple MS

Once you have loaded data into measurement sets on disk, you can use the `concat` task to combine them. Currently, `concat` will add a second MS to an existing MS (not producing a new one). This would be run multiple times if you had more than two sets to combine.

See § 2.2.9 for details.

1.5.2 Data Examination, Editing, and Flagging

The main data examination and flagging tasks are:

- `listobs` — summarize the contents of a MS (§ 2.2.5);
- `flagmanager` — save and manage versions of the flagging entries in the measurement set (§ 3.2);
- `flagautocorr` — non-interactive flagging of auto-correlations (§ 3.3);
- `plotms` — interactive X-Y plotting and flagging of visibility data (§ 3.4.1);
- (`plotxy` — interactive X-Y plotting and flagging of visibility data (§ 3.4.2), **note:** `plotxy` is slower than `plotms` and will eventually be phased out, `plotxy` is still useful to create scripted hardcopy output, this functionality will likely be available in `plotms` in the next release);
- `flagdata` — non-interactive flagging (and unflagging) of specified data (§ 3.5);
- `viewer` — the CASA viewer can display (as a raster image) MS data, with some editing capabilities (§ 7);

These tasks allow you to list, plot, and/or flag data in a CASA MS.

There will eventually be tasks for “automatic” flagging to data based upon statistical criteria. Stay tuned.

Examination and editing of synthesis data is described in Chapter 3.

Visualization and editing of an MS using the `casaviewer` is described in Chapter 7.

1.5.2.1 Interactive X-Y Plotting and Flagging

The principal tool for making X-Y plots of visibility data is `plotxy` (see § 3.4.2). Amplitudes and phases (among other things) can be plotted against several x-axis options.

Interactive flagging (i.e., “see it – flag it”) is possible on the `plotxy` X-Y displays of the data (§ 3.4.2.10). Since flags are inserted into the measurement set, it is useful to backup (or make a copy) of the current flags before further flagging is done, using `flagmanager` (§ 3.2). Copies of the flag table can also be restored to the MS in this way.

1.5.2.2 Flag the Data Non-interactively

The `flagdata` task (§ 3.5) will flag the visibility data set based on the specified data selections. The `listobs` task (§ 2.2.5) may be run (e.g. with `verbose=True`) to provide some of the information needed to specify the flagging scope.

1.5.2.3 Viewing and Flagging the MS

The CASA `viewer` can be used to display the data in the MS as a (grayscale or color) raster image. The MS can also be edited. Use of the `viewer` on an MS is detailed in § 7.4.

1.5.3 Calibration

The major calibration tasks are:

- `setjy` — Computes the model visibilities for a specified source given a flux density or model image, knows about standard calibrator sources (§ 4.3.4);
- `gencal` — Creates a calibration table for known delay and antenna position offsets (§ 4.3.5);
- `bandpass` — Solves for frequency-dependent (bandpass) complex gains (§ 4.4.2);
- `gaincal` — Solves for time-dependent (frequency-independent) complex gains (§ 4.4.3);
- `fluxscale` — Bootstraps the flux density scale from standard calibrators (§ 4.4.4);
- `polcal` — polarization calibration (§ 4.4.5);
- `applycal` — Applies calculated calibration solutions (§ 4.6.1);
- `clearcal` — Re-initializes calibrated visibility data in a given measurement set (§ 4.6.3);
- `listcal` — Lists calibration solutions (§ 4.5.2);
- `plotcal` — Plots (and optionally flags) calibration solutions (§ 4.5.1);
- `uvcontsub` — carry out uv-plane continuum subtraction for spectral-line data (§ 4.7.4);
- `split` — write out a new (calibrated) MS for specified sources (§ 4.7.1);
- `cvel` — Regrid a spectral MS onto a new frequency channel system (§ 4.7.5).

During the course of calibration, the user will specify a set of calibrations to pre-apply before solving for a particular type of effect, for example gain or bandpass or polarization. The solutions are stored in a calibration table (subdirectory) which is specified by the user, *not* by the task: care must be taken in naming the table for future use. The user then has the option, as the calibration process proceeds, to accumulate the current state of calibration in a new cumulative table. Finally, the calibration can be applied to the dataset.

Synthesis data calibration is described in detail in Chapter 4.

1.5.3.1 Prior Calibration

The `setjy` task places the Fourier transform of a standard calibration source model in the `MODEL_DATA` column of the measurement set. This can then be used in later calibration tasks. Currently, `setjy` knows the flux density as a function of frequency for several standard VLA flux calibrators, and the value of the flux density can be manually inserted for any other source. If the source is not well-modeled as a point source, then a model image of that source structure can be used (with the total flux density scaled by the values given or calculated above for the flux density). Models are provided for the standard VLA calibrators.

Antenna gain-elevation curves (e.g. for the VLA antennas) and atmospheric optical depth corrections (applied as an elevation-dependent function) may be pre-applied before solving for the bandpass and gains. This is currently done by setting the `gaincurve` and `opacity` parameters in the various calibration solving tasks.

There is now a `gencal` task to create calibration tables for known delay and antenna position offsets.

See § 4.3 for more details.

1.5.3.2 Bandpass Calibration

The `bandpass` task calculates a bandpass calibration solution: that is, it solves for gain variations in frequency as well as in time. Since the bandpass (relative gain as a function of frequency) generally varies much more slowly than the changes in overall (mean) gain solved for by `gaincal`, one generally uses a long time scale when solving for the bandpass. The default 'B' solution mode solves for the gains in frequency slots consisting of channels or averages of channels.

A polynomial fit for the solution (solution type 'BPOLY') may be carried out instead of the default frequency-slot based 'B' solutions. This single solution will span (combine) multiple spectral windows.

Bandpass calibration is discussed in detail in § 4.4.2.

If the gains of the system are changing over the time that the bandpass calibrator is observed, then you may need to do an initial gain calibration (see next step).

1.5.3.3 Gain Calibration

The `gaincal` task determines solutions for the time-based complex antenna gains, for each spectral window, from the specified calibration sources. A solution interval may be specified. The default 'G' solution mode solved for gains in specified time solution intervals.

A spline fit for the solution (solution type 'GSPLINE') may be carried out instead of the default time-slot based 'G' solutions. This single solution will span (combine) multiple spectral windows.

See § 4.4.3 for more on gain calibration.

1.5.3.4 Polarization Calibration

The `polcal` task will solve for any unknown polarization leakage and cross-hand phase terms ('D' and 'X' solutions). The 'D' leakage solutions will work on sources with no polarization, sources with known (and supplied) polarization, and sources with unknown polarization tracked through a range in parallactic angle on the sky.

The solution for the unknown cross-hand polarization phase difference 'X' term requires a polarized source with known linear polarization (Q,U).

See § 4.4.5 for more on polarization calibration.

1.5.3.5 Examining Calibration Solutions

The `plotcal` task (§ 4.5.1) will plot the solutions in a calibration table. The `xaxis` choices include time (for `gaincal` solutions) and channel (e.g. for `bandpass` calibration). The `plotcal` interface and plotting surface is similar to that in `plotxy`. Eventually, `plotcal` will allow you to flag and unflag calibration solutions in the same way that data can be edited in `plotxy`.

The `listcal` task (§ 4.5.2) will print out the calibration solutions in a specified table.

1.5.3.6 Bootstrapping Flux Calibration

The `fluxscale` task bootstraps the flux density scale from “primary” standard calibrators to the “secondary” calibration sources. Note that the flux density scale must have been previously established on the “primary” calibrator(s), typically using `setjy`, and of course a calibration table containing valid solutions for all calibrators must be available.

See § 4.4.4 for more.

1.5.3.7 Correcting the Data

The final step in the calibration process, `applycal` may be used to apply several calibration tables (e.g., from `gaincal` or `bandpass`). The corrections are applied to the `DATA` column of the visibility, writing the `CORRECTED_DATA` column which can then be plotted (e.g. in `plotxy`), `split` out as the `DATA` column of a new MS, or imaged (e.g. using `clean`). Any existing corrected data are overwritten.

See § 4.6.1 for details.

1.5.3.8 Splitting the Data

After a suitable calibration is achieved, it may be desirable to create one or more new measurement sets containing the data for selected sources. This can be done using the `split` task (§ 4.7.1).

Further imaging and calibration (e.g. self-calibration) can be carried out on these split Measurement Sets.

1.5.3.9 UV Continuum subtraction

For spectral line data, continuum subtraction can be performed in the image domeain (`imcontsub`) or in the uv domain. For the latter, there are two tasks available: `uvcontsub` subtracts polynomial of desired order from each baseline, defined by line-free channels. `uvcontsub2` performs this process (as of release 3.0.1 only first order) across spectral windows in a broad band dataset.

1.5.3.10 Transforming the Data to a new frame

If you want to transform your dataset to a different frequency and velocity frame than the one it was observed in, then you can use the `cvel` task (§ 4.7.5). Alternatively, you can do the regridding during the imaging process in `clean` without running `cvel` before.

1.5.4 Synthesis Imaging

The key synthesis imaging tasks are:

- `clean` — Calculates a deconvolved image based on the visibility data, using one of several `clean` algorithms (§ 5.3);
- `feather` — Combines a single dish and synthesis image in the Fourier plane (§ 5.4).

Most of these tasks are used to take calibrated interferometer data, with the possible addition of a single-dish image, and reconstruct a model image of the sky. **Alert:** The `clean` task is now even more powerful and incorporates the functionality of previous specialized tasks such as `mosaic` and `widefield`.

See Chapter 5 for more on synthesis imaging.

1.5.4.1 Cleaning a single-field image or a mosaic

The CLEAN algorithm is the most popular and widely-studied method for reconstructing a model image based on interferometer data. It iteratively removes at each step a fraction of the flux in the brightest pixel in a defined region of the current “dirty” image, and places this in the model image. The `clean` task implements the CLEAN algorithm for single-field data. The user can choose from a number of options for the particular flavor of CLEAN to use.

Often, the first step in imaging is to make a simple gridded Fourier inversion of the calibrated data to make a “dirty” image. This can then be examined to look for the presence of noticeable emission above the noise, and to assess the quality of the calibration by searching for artifacts in the image. This is done using `clean` with `niter=0`.

The `clean` task can jointly deconvolve mosaics as well as single fields, and also has options to do wide-field and wide-band multi-frequency synthesis imaging.

See § 5.3 for an in-depth discussion of the `clean` task.

1.5.4.2 Feathering in a Single-Dish image

If you have a single-dish image of the large-scale emission in the field, this can be “feathered” in to the image obtained from the interferometer data. This is carried out using the `feather` tasks as the weighted sum in the uv-plane of the gridded transforms of these two images. While not as accurate as a true joint reconstruction of an image from the synthesis and single-dish data together, it is sufficient for most purposes.

See § 5.4 for details on the use of the `feather` task.

1.5.5 Self Calibration

Once a calibrated dataset is obtained, and a first deconvolved model image is computed, a “self-calibration” loop can be performed. Effectively, the model (not restored) image is passed back to another calibration process (on the target data). This refines the calibration of the target source, which up to this point has had (usually) only external calibration applied. This process follows the regular calibration procedure outlined above.

Any number of self-calibration loops can be performed. As long as the images are improving, it is usually prudent to continue the self-calibration iterations.

This process is described in § 5.8.

1.5.6 Data and Image Analysis

The key data and image analysis tasks are:

- `imhead` — summarize and manipulate the “header” information in a CASA image (§ 6.2);
- `imcontsub` — perform continuum subtraction on a spectral-line image cube (§ 6.3);
- `immath` — perform mathematical operations on or between images (§ 6.5);
- `immoments` — compute the moments of an image cube (§ 6.6);
- `imstat` — calculate statistics on an image or part of an image (§ 6.7);
- `imval` — extract values of one or more pixels, as a spectrum for cubes, from an image (§ 6.8);
- `imfit` — simple 2D Gaussian fitting of single components to a region of an image (§ 6.4);
- `imregrid` — regrid an image onto the coordinate system of another image (§ 6.9);

- **viewer** — there are useful region statistics and image cube plotting capabilities in the viewer (§ 7).

1.5.6.1 What’s in an image?

The `imhead` task will print out a summary of image “header” keywords and values. This task can also be used to retrieve and change the header values.

See § 6.2 for more.

1.5.6.2 Image statistics

The `imstat` task will print image statistics. There are options to restrict this to a box region, and to specified channels and Stokes of the cube. This task will return the statistics in a Python dictionary return variable.

See § 6.7 for more.

1.5.6.3 Image values

The `imval` task will values from an image. There are options to restrict this to a box region, and to return specified channels and Stokes of the cube as a spectrum. This task will return these values in a Python dictionary return variable which can then be operated on in the `casapy` environment.

See § 6.7 for more.

1.5.6.4 Moments of an image cube

The `immoments` task will compute a “moments” image of an input image cube. A number of options are available, from the traditional true moments (zero, first, second) and variations thereof, to other images such as median, minimum, or maximum along the moment axis.

See § 6.6 for details.

1.5.6.5 Image math

The `immath` task will allow you to form a new image by mathematical combinations of other images (or parts of images). This is a powerful, but tricky, task to use.

See § 6.5 for more.

1.5.6.6 Regridding an Image

It is occasionally necessary to regrid an image onto a new coordinate system. The `imregrid` task can be used to regrid an input image onto the coordinate system of an existing template image, creating a new output image.

See § 6.9 for a description of this task.

1.5.6.7 Displaying Images

To display an image use the `viewer` task. The viewer will display images in raster, contour, or vector form. Blinking and movies are available for spectral-line image cubes. To start the viewer, type:

```
viewer
```

Executing the `viewer` task will bring up two windows: a viewer screen showing the data or image, and a file catalog list. Click on an image or ms from the file catalog list, choose the proper display, and the image should pop up on the screen. Clicking on the wrench tool (second from left on upper left) will obtain the data display options. Most functions are self-documenting.

The viewer can be run outside of casapy by typing `casaviewer`.

See § 7 for more on viewing images.

1.5.7 Getting data and images out of CASA

The key data and image export tasks are:

- `exportuvfits` — export a CASA MS in UVFITS format (§ 2.2.3);
- `exportfits` — export a CASA image table as FITS (§ 6.11).

These tasks can be used to export a CASA MS or image to UVFITS or FITS respectively. See the individual sections referred to above for more on each.

Chapter 2

Visibility Data Import, Export, and Selection

To use CASA to process your data, you first will need to get it into a form that is understood by the package. These are “measurement sets” for synthesis (and single dish) data, and “image tables” for images.

There are a number of tasks used to fill telescope-specific data, to import/export standard formats, to list data contents, and to concatenate multiple datasets. These are:

- `importasdm` — import ALMA or EVLA data in ASDM format (§ 2.2.1)
- `importuvfits` — import visibility data in UVFITS format (§ 2.2.3.1)
- `importvla` — import data from VLA that is in *export* format (§ 2.2.2)
- `exportuvfits` — export a CASA MS in UVFITS format (§ 2.2.3.2)
- `listobs` — summarize the contents of a MS (§ 2.2.5)
- `listvis` — list the data in a MS (§ 2.2.6)
- `vishead` — list and change the metadata contents of a MS (§ 2.2.7)
- `visstat` — statistics on data in a MS (§ 2.2.8)
- `concat` — concatenate two or more MS into a new MS (§ 2.2.9)

In CASA, there is a standard syntax for selection of data that is employed by multiple tasks. This is described in § 2.3.

There are also tasks for the import and export of image data using FITS:

- `importfits` — import a FITS image into a CASA *image* format table (§ 6.11)
- `exportfits` — export a CASA image table as FITS (§ 6.11)

2.1 CASA Measurement Sets

Data is handled in CASA via the `table` system. In particular, visibility data are stored in a CASA table known as a Measurement Set (MS). Details of the physical and logical MS structure are given below, but for our purposes here an MS is just a construct that contains the data. An MS can also store single dish data (essentially a set of auto-correlations of a 1-element interferometer), though there are also data formats more suitable for single-dish spectra (see § A).

Note that images are handled through special `image` tables, although standard FITS I/O is also supported. Images and `image` data are described in a separate chapter.

Unless your data was previously processed by CASA or software based upon its predecessor `aips++`, you will need to import it into CASA as an MS. Supported formats include some “standard” flavors of UVFITS, the VLA “Export” archive format, and most recently, the ALMA Science Data Model (ASDM) format. These are described below in § 2.2.

Once in Measurement Set form, your data can be accessed through various tools and tasks with a common interface. The most important of these is the *data selection interface* (§ 2.3) which allows you to specify the subset of the data on which the tasks and tools will operate.

Inside the Toolkit:
Measurement sets are handled in the `ms` tool. Import and export methods include `ms.fromfits` and `ms.tofits`.

2.1.1 Under the Hood: Structure of the Measurement Set

It is not necessary that a casual CASA user know the specific details on how the data in the MS is stored and the contents of all the sub-tables. However, we will occasionally refer to specific “columns” of the MS when describing the actions of various tasks, and thus we provide the following synopsis to familiarize the user with the necessary nomenclature. You may skip ahead to subsequent sections if you like!

Inside the Toolkit:
Generic CASA tables are handled in the `tb` tool. You have direct access to keywords, rows and columns of the tables with the methods of this tool.

All CASA data files, including Measurement Sets, are written into the current working directory by default, with each CASA table represented as a separate sub-directory. MS names therefore need only comply with UNIX file or directory naming conventions, and can be referred to from within CASA directly, or via full path names.

An MS consists of a `MAIN` table containing the visibility data. and associated sub-tables containing auxiliary or secondary information. The tables are logical constructs, with contents located in the physical `table.*` files on disk. The `MAIN` table consists of the `table.*` files in the main directory of the `ms`-file itself, and the other tables are in the respective subdirectories. The various MS tables and sub-tables can be seen by listing the contents of the MS directory itself (e.g. using Unix `ls`), or via the `browsetable` task (§ 3.6).

See Fig 2.1 for an example of the contents of a MS directory. Or, from the `casapy` prompt,

```

CASA <1>: ls ngc5921.ms
IPython system call: ls -F ngc5921.ms
ANTENNA          POLARIZATION      table.f1          table.f3_TSM1    table.f8
DATA_DESCRIPTION PROCESSOR          table.f10         table.f4          table.f8_TSM1
FEED             SORTED_TABLE      table.f10_TSM1   table.f5          table.f9
FIELD           SOURCE            table.f11         table.f5_TSM1    table.f9_TSM1
FLAG_CMD        SPECTRAL_WINDOW  table.f11_TSM1   table.f6          table.info
HISTORY         STATE            table.f2          table.f6_TSM0    table.lock
OBSERVATION     table.dat        table.f2_TSM1    table.f7
POINTING        table.f0         table.f3          table.f7_TSM1

```

Note that the MAIN table information is contained in the `table.*` files in this directory. Each of the sub-table sub-directories contain their own `table.dat` and other files, e.g.

```

CASA <2>: ls ngc5921.ms/SOURCE
IPython system call: ls -F ngc5921.ms/SOURCE
table.dat table.f0 table.f0i table.info table.lock

```

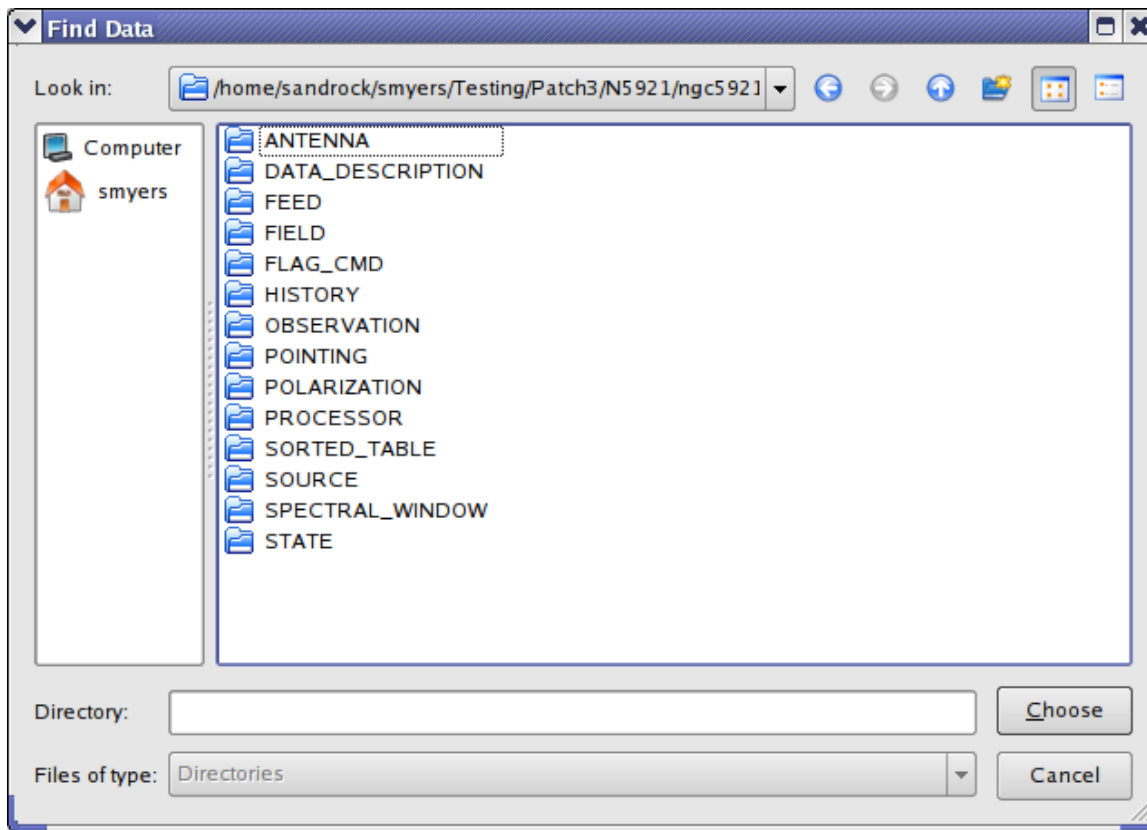


Figure 2.1: The contents of a Measurement Set. These tables compose a Measurement Set named `ngc5921.demo.ms` on disk. This display is obtained by using the **File:Open** menu in `browsetable` and left double-clicking on the `ngc5921.demo.ms` directory.

Each “row” in a table contains entries for a number of specified “columns”. For example, in the `MAIN` table of the MS, the original visibility data is contained in the `DATA` column — each “cell” contains a matrix of observed complex visibilities for that row at a single time stamp, for a single baseline in a single spectral window. The shape of the data matrix is given by the number of channels and the number of correlations (voltage-products) formed by the correlator for an array.

Table 2.1 lists the non-data columns of the `MAIN` table that are most important during a typical data reduction session. Table 2.2 lists the key data columns of the `MAIN` table of an interferometer MS. The MS produced by fillers for specific instruments may insert special columns, such as `ALMA_PHASE_CORR`, `ALMA_NO_PHAS_CORR` and `ALMA_PHAS_CORR_FLAG_ROW` for ALMA data filled using the `importasdm` filler (§ 2.2.1). These columns are visible in `browsetable` and are accessible from the toolkit in the `ms` tool (e.g. the `ms.getdata` method) and from the `tb` “table” tool (e.g. using `tb.getcol`).

Note that when you examine table entries for IDs such as `FIELD_ID` or `DATA_DESC_ID`, you will see 0-based numbers.

Table 2.1: Common columns in the `MAIN` table of the MS.

Parameter	Contents
<code>ANTENNA1</code>	First antenna in baseline
<code>ANTENNA2</code>	Second antenna in baseline
<code>FIELD_ID</code>	Field (source no.) identification
<code>DATA_DESC_ID</code>	Spectral window number, polarization identifier pair (IF no.)
<code>ARRAY_ID</code>	Subarray number
<code>OBSERVATION_ID</code>	Observation identification
<code>POLARIZATION_ID</code>	Polarization identification
<code>SCAN_NUMBER</code>	Scan number
<code>TIME</code>	Integration midpoint time
<code>UVW</code>	UVW coordinates

The MS can contain a number of “scratch” columns, which are used to hold useful versions of other columns such as the data or weights for further processing. The most common scratch columns are:

- `CORRECTED_DATA` — used to hold calibrated data for imaging or display;
- `MODEL_DATA` — holds the Fourier inversion of a particular model image for calibration or imaging;
- `IMAGING_WEIGHT` — holds the gridding weights to be used in imaging.

The creation and use of the scratch columns is generally done behind the scenes, but you should be aware that they are there (and when they are used).

Table 2.2: Commonly accessed MAIN Table data-related columns. Note that the columns ALMA_PHASE_CORR, ALMA_NO_PHAS_CORR and ALMA_PHAS_CORR_FLAG_ROW are specific to ALMA data filled using the `importasdm` filler.

Column	Format	Contents
DATA	Complex(N_c, N_f)	complex visibility data matrix (= ALMA_PHASE_CORR by default)
FLAG	Bool(N_c, N_f)	cumulative data flags
WEIGHT	Float(N_c)	weight for a row
WEIGHT_SPECTRUM	Float(N_c, N_f)	individual weights for a data matrix
ALMA_PHASE_CORR	Complex(N_c, N_f)	on-line phase corrected data (<i>Not in VLA data</i>)
ALMA_NO_PHAS_CORR	Bool(N_c, N_f)	data that has not been phase corrected (<i>Not in VLA data</i>)
ALMA_PHAS_CORR_FLAG_ROW	Bool(N_c, N_f)	flag to use phase-corrected data or not (<i>not in VLA data</i>)
MODEL_DATA	Complex(N_c, N_f)	Scratch: created by calibrator or imager tools
CORRECTED_DATA	Complex(N_c, N_f)	Scratch: created by calibrator or imager tools
IMAGING_WEIGHT	Float(N_c)	Scratch: created by calibrator or imager tools

The most recent specification for the MS is **Aips++ MeasurementSet definition version 2.0** (<http://casa.nrao.edu/Memos/229.html>).

2.2 Data Import and Export

There are a number of tasks available to bring data in various forms into CASA as a Measurement Set:

- ALMA and EVLA Science Data Model format data can be imported into CASA (`importasdm`)
- VLA Archive format data can be imported into CASA (`importvla`)
- UVFITS format can be imported into and exported from CASA (`importuvfits` and `exportuvfits`)

2.2.1 ALMA & EVLA: Filling Science Data Model (ASDM) data

The ALMA and EVLA projects have agreed upon a common archival science data model (ASDM) format, and have jointly developed the software to fill this data into CASA.

Under the Hood:

The `importasdm` task is just an interface to the stand-alone `asdm2MS` application. To find out the command-line arguments to this application, do `asdm2MS --help`.

In the ASDM format, the bulk of the data is contained in large binary data format (BDF) tables, with the meta-data and ancillary information in XML tables. This is structured as a directory, like the MS, and was designed to be similar to the MS to facilitate conversion.

The `importasdm` task will fill SDM1 format data (Oct 2009 or later) into a CASA visibility data set (MS). The default inputs are:

```
# importasdm :: Convert an Science Data Model observation into a CASA visibility file
asdm          = ''      # Name of input asdm directory (on disk)
vis           = ''      # Root name of the ms to be created. Note the .ms is added
singledish    = False   # Set true to output single-dish data format
corr_mode     = 'all'   # correlation mode to be input (ao, co, ac, all)
srt           = 'all'   # spectral resolution type (fr, ca, bw, or all)
time_sampling = 'all'   # time sampling (INTEGRATION and/or SUBINTEGRATION) (i, si, or all)
ocorr_mode    = 'ca'    # output correlation mode AUTO_ONLY (ao), CROSS_ONLY (co), CROSS_AND_AUTO
compression   = False   # Flag for turning on data compression
asis          = ''      # string of table names separated by spaces to be passed verbatim (* all)
wvr_corrected_data = 'no' # 'no' (default), 'yes'= WVR corrected, 'both'= output separate MS
verbose       = False   # Output lots of information while the filler is working
overwrite     = False   # Overwrite an existing MS
showversion   = False   # Report the version of asdm2MS being used
async        = False   # If true the taskname must be started using importasdm(...)
```

ALERT: Currently there are no options for filling selected data (you get the whole data set).

If `singledish=True`, output data format is scantable (single-dish data format, see A) instead of MS. In that case, you must specify name or id of the antenna that you want to obtain data. This can be done by using antenna parameter that is defined as a subparameter of `singledish`. For single-dish mode, only auto-correlation data are filled, i.e. `ocorr_mode` is forcibly set to 'ao'.

2.2.1.1 Filling SDM v0 data using `importoldasdm`

If you are filling old test data taken before October 2009 in the SDM0 format, then you should use the `importoldasdm` task to fill this.

```
# importoldasdm :: Convert a Science Data Model v0 observation into a CASA visibility file
asdm          = ''      # Name of input asdm directory (on disk)
corr_mode     = 'all'   # correlation mode to be considered on input
srt           = 'all'   # specifies the spectral resolution type to be considered on input
time_sampling = 'all'   # time sampling (INTEGRATION and/or SUBINTEGRATION)
ocorr_mode    = 'co'    # AUTO_ONLY (ao) or CROSS_ONLY (co) or CROSS_AND_AUTO (ca)
compression   = False   # Flag for turning on data compression
async        = False   # If true the taskname must be started using importoldasdm(...)
```

2.2.2 VLA: Filling data from archive format (importvla)

VLA data in archive format (i.e., as downloaded from the VLA data archive) are read into CASA from disk using the `importvla` task. The inputs are:

```
# importvla :: import VLA archive file(s) to a measurement set:

archivefiles =      '' # Name of input VLA archive file(s)
vis           =      '' # Name of output visibility file
bandname      =      '' # VLA frequency band name:''=>obtain all bands in archive files
frequencytol  = 150000.0 # Frequency shift to define a unique spectral window (Hz)
project       =      '' # Project name: '' => all projects in file
starttime     =      '' # start time to search for data
stoptime      =      '' # end time to search for data
applysys      =      True # apply nominal sensitivity scaling to data & weights
autocorr      =      False # import autocorrelations to ms, if set to True
antnamescheme = 'new'   # 'old' or 'new'; 'VA04' or '4' for ant 4
keepblanks    =      False # Fill scans with empty source names (e.g. tipping scans)?
evlabands     =      False # Use updated eVLA frequencies and bandwidths
async         =      False
```

The main parameters are `archivefiles` to specify the input VLA Archive format file names, and `vis` to specify the output MS name.

ALERT: The scaling of VLA data both before and after the June 2007 Modcomp-turnoff is fully supported, based on the value of `applysys`.

The NRAO Archive is located at:

- <https://archive.nrao.edu>

Note that `archivefiles` takes a string or list of strings, as there are often multiple files for a project in the archive.

For example:

```
archivefiles = ['AP314_A950519.xp1', 'AP314_A950519.xp2']
vis = 'NGC7538.ms'
```

The `importvla` task allows selection on the frequency band. Suppose that you have 1.3 cm line observations in K-band and you have copied the archive data files `AP314.A95019.xp*` to your working directory and started `casapy`. Then,

```
default('importvla')
archivefiles = ['AP314_A950519.xp1', 'AP314_A950519.xp2', 'AP314_A950519.xp3']
vis = 'ngc7538.ms'
bandname = 'K'
frequencytol = 10e6
importvla()
```

If the data is located in a different directory on disk, then use the full path name to specify each archive file, e.g.:

```
archivefiles=['/home/rohir2/jmcmulli/ALMATST1/Data/N7538/AP314_A950519.xp1',\
             '/home/rohir2/jmcmulli/ALMATST1/Data/N7538/AP314_A950519.xp2',\
             '/home/rohir2/jmcmulli/ALMATST1/Data/N7538/AP314_A950519.xp3']
```

Important Note: `importvla` will import the on-line flags (from the VLA system) along with the data. Shadowed antennas will also be flagged. The flags will be put in the `MAIN` table and thus available to subsequent tasks and tools. If you wish to revert to unflagged data, use `flagmanager` (§ 3.2) to save the flags (if you wish), and then use `flagdata` (§ 3.5) with `mode='manualflag'` and `unflag=True` to toggle off the flags.

The other parameters are:

2.2.2.1 Parameter `applytys`

The `applytys` parameter controls whether the nominal sensitivity scaling (based on the measured `TSYS`, with the weights scaled accordingly using the integration time) is applied to the visibility amplitudes or not. If `True`, then it will be scaled so as to be the same as `AIPS FILLM` (ie. approximately in deciJanskys). Note that post-Modcomp data is in raw correlation coefficient and will be scaled using the `TSYS` values, while Modcomp-era data had this applied online. In all cases `importvla` will do the correct thing to data and weights based on an internal flag in the VLA Archive file, either scaling it or unscaling based on your choice for `applytys`.

If `applytys=True` and you see strange behavior in data amplitudes, it may be due to erroneous `TSYS` values from the online system. You might want to then fill with `applytys=False` and look at the correlation coefficients to see if the behavior is as expected.

2.2.2.2 Parameter `bandname`

The `bandname` indicates the VLA Frequency band(s) to load, using the traditional bandname codes. These are:

- '4' = 48-96 MHz
- 'P' = 298-345 MHz
- 'L' = 1.15-1.75 GHz
- 'C' = 4.2-5.1 GHz
- 'X' = 6.8-9.6 GHz
- 'U' = 13.5-16.3 GHz
- 'K' = 20.8-25.8 GHz

- 'Q' = 38-51 GHz
- '' = all bands (default)

Note that as the transition from the VLA to EVLA progresses, the actual frequency ranges covered by the bands will expand, and additional bands will be added (namely 'S' from 1-2 GHz and 'A' from 26.4-40 GHz).

2.2.2.3 Parameter `frequencytol`

The `frequencytol` parameter specifies the frequency separation tolerated when assigning data to spectral windows. The default is `frequencytol=150000` (Hz). For Doppler tracked data, where the sky frequency changes with time, a `frequencytol < 10000` Hz may produce too many unnecessary spectral windows.

2.2.2.4 Parameter `project`

You can specify a specific `project` name to import from archive files. The default '' will import data from all projects in file(s) `archivefiles`.

For example for VLA Project AL519:

```
project = 'AL519'    # this will work
project = 'al519'   # this will also work
```

while `project='AL0519'` will NOT work (even though that is what queries to the VLA Archive will print it as - sorry!).

2.2.2.5 Parameters `starttime` and `stoptime`

You can specify start and stop times for the data, e.g.:

```
starttime = '1970/1/31/00:00:00'
stoptime  = '2199/1/31/23:59:59'
```

Note that the blank defaults will load all data fitting other criteria.

2.2.2.6 Parameter `autocorr`

Note that autocorrelations are filled into the data set if `autocorr=True`. Generally for the VLA, autocorrelation data is not useful, and furthermore the imaging routine will try to image the autocorrelation data (it assumes it is single dish data) which will swamp any real signal. Thus, if you do fill the autocorrelations, you will have to flag them before imaging.

2.2.2.7 Parameter antnamescheme

The `antnamescheme` parameter controls whether `importvla` will try to use a naming scheme where EVLA antennas are prefixed with `EA` (e.g. `'EA16'`) and old VLA antennas have names prefixed with `VA` (e.g. `'VA11'`). Our method to detect whether an antenna is EVLA is not yet perfected, and thus unless you require this feature, simply use `antnamescheme='old'`.

2.2.2.8 Parameter evlabands

The `evlabands=True` option is provided to allow users to access EVLA frequencies outside the standard VLA tunings (e.g. the extended C-band above 6 GHz). **ALERT:** use of this option for standard VLA data will cause unexpected associations, such as X-band data below 8 GHz being extracted to C-band (as the EVLA C-band is 4–8 GHz). Use with care.

2.2.3 UVFITS Import and Export

The UVFITS format is not exactly a standard, but is a popular archive and transport format nonetheless. CASA supports UVFITS files written by the AIPS FITTP task, and others.

UVFITS is supported for both import and export.

2.2.3.1 Import using importuvfits

To import UVFITS format data into CASA, use the `importuvfits` task:

```
CASA <1>: inp(importuvfits)
fitsfile      =      '' # Name of input UVFITS file
vis           =      '' # Name of output visibility file (MS)
antnamescheme =      'old' # For VLA only; 'new' or 'old'; 'VA04' or '04' for VLA ant 4
async        =      False # if True run in the background, prompt is freed
```

This is straightforward, since all it does is read in a UVFITS file and convert it as best it can into a MS.

For example:

```
importuvfits(fitsfile='NGC5921.fits',vis='ngc5921.ms')
```

ALERT: We cannot currently fill CARMA data exported via Miriad UVFITS.

2.2.3.2 Export using exportuvfits

The `exportuvfits` task will take a MS and write it out in UVFITS format. The defaults are:

```
# exportuvfits :: Convert a CASA visibility data set (MS) to a UVFITS file

vis          =      ''      # Name of input visibility file
fitsfile     =      ''      # Name of output UVFITS file)
datacolumn   = 'corrected'  # which data to write (data, corrected, model)
field        =      ''      # Field name list
spw          =      ''      # Spectral window and channel selection
antenna      =      ''      # antenna list to select
time         =      ''      # time range selection
nchan        =      -1      # Number of channels to select
start        =      0       # Start channel
width        =      1       # Channel averaging width (value>1 indicates averaging)
writesyscal  =      False   # Write GC and TY tables
multisource  =      True    # Write in multi-source format
combinespw   =      True    # Combine spectral windows (True for AIPS)
writestation =      True    # Write station name instead of antenna name
async       =      False   # if True run in the background, prompt is freed
```

For example:

```
exportuvfits(vis='ngc5921.split.ms',
             fitsfile='NGC5921.split.fits',
             multisource=False)
```

The MS selection parameters `field`, `spw`, `antenna`, and `timerange` follow the standard selection syntax described in § 2.3.

ALERT: The `nchan`, `start`, and `width` parameters will be superseded by channel selection in `spw`. Currently, there is a `time` parameter rather than `timerange`.

The `datacolumn` parameter chooses which data-containing column of the MS (see § 2.1.1) is to be written out to the UV FITS file. Choices are: `'data'`, `'corrected'`, and `'model'`.

There are a number of special parameters that control what is written out. These are mostly here for compatibility with AIPS.

The `writesyscal` parameter toggles whether GC and TY extension tables are written. These are important for VLBA data, and for EVLA data. **ALERT:** Not yet available.

The `multisource` parameter determines whether the UV FITS file is a multi-source file or a single-source file, if you have a *single-source* MS or choose only a single source. Note: the difference between a single-source and multi-source UVFITS file here is whether it has a source (SU) table and the source ID in the random parameters. Some programs (i.e. `difmap`) only accept single-source files. If you select more than one source in fields, then the `multisource` parameter will be overridden to be True regardless.

The `combinespw` parameter allows, if some conditions are met, exporting all of spectral windows (SpW) as a set of "IF"s in a single "FREQID" setup instead of giving each SpW its own FREQID in the FITS file. In this context an IF (Intermediate Frequency) is a specialization of an SpW, where each IF in a UV FITS file must have the same number of channels and polarizations, each channel must have the same width, and each IF must be present (even if flagged) throughout the entire observation. If these conditions are not met the data must be exported using multiple FREQIDs, the UV FITS equivalent of a general SpW. This matters since many (sub)programs will work with multiple IFs, but not multiple FREQIDs. For example, a UV FITS file with multiple FREQIDs can be read by AIPS, but you may find that you have to separate the FREQIDs with `SPLIT` before you can do very much with them. Therefore `combinespw=True` should be `True` if possible. Typically MSes where each band was observed simultaneously can be exported with `combinespw=True`. MSes where the tuning changed with time, e.g. 10 minutes at 4.8 GHz followed by 15 minutes at 8.4 GHz, should be exported to multiple UV FITS files using `spw` to select one tuning (set of simultaneous SpWs) per file.

The `multisource` parameter determines whether the UV FITS file is a multi-source file or a single-source file, if you have a *single-source* MS or choose only a single source. Note: the difference between a single-source and multi-source UVFITS file here is whether it has a source (SU) table and the source ID in the random parameters. If you select more than one source in `fields`, then the `multisource` parameter will be overridden to be `True` regardless.

The `combinespw` parameter allows combination of all spectral windows at one time. If `True`, then all spectral windows must have the same shape. For AIPS to read an exported file, then set `combinespw=True`.

The `writestation` parameter toggles the writing of the station name instead of antenna name.

2.2.4 Handling Measurement Set metadata and data

There are tasks provided for basic listing and manipulation of Measurement Set data and metadata. These include:

- `listobs` — summarize the contents of a MS (§ 2.2.5)
- `vishead` — list and change the metadata contents of a MS (§ 2.2.7)
- `visstat` — statistics on data in a MS (§ 2.2.8)
- `concat` — concatenate two or more MS into a new MS (§ 2.2.9)

2.2.5 Summarizing your MS (`listobs`)

Once you import your data into a CASA Measurement Set, you can get a summary of the MS contents with the `listobs` task.

The inputs are:

```
vis           =      ''           # Name of input visibility file (MS)
verbose      =      True          # Extended summary list of data set in logger
```

The summary will be written to the logger and to the casapy.log file. For example,

```
listobs('n5921.ms')
```

results in the logger messages:

```
=====
MeasurementSet Name: /home/sandrock3/smyers/Testing3/Patch4/N5921/ngc5921.demo.ms
MS Version 2
=====
Observer: TEST      Project:
Observation: VLA
Data records: 22653      Total integration time = 5280 seconds
Observed from 13-Apr-1995/09:19:00.0 to 13-Apr-1995/10:47:00.0 (TAI)

ObservationID = 0      ArrayID = 0
Date      Timerange (TAI)      Scan  FldId  FieldName      nVis  Int(s)  SpwIds
13-Apr-1995/09:19:00.0 - 09:24:30.0      1      0 1331+305000* 4509  30      [0]
                09:27:30.0 - 09:29:30.0      2      1 1445+099000* 1890  30      [0]
                09:33:00.0 - 09:48:00.0      3      2 N5921_2      6048  30      [0]
                09:50:30.0 - 09:51:00.0      4      1 1445+099000* 756   30      [0]
                10:22:00.0 - 10:23:00.0      5      1 1445+099000* 1134  30      [0]
                10:26:00.0 - 10:43:00.0      6      2 N5921_2      6804  30      [0]
                10:45:30.0 - 10:47:00.0      7      1 1445+099000* 1512  30      [0]
      (nVis = Total number of time/baseline visibilities per scan)
Fields: 3
ID  Code Name      RA      Decl      Epoch  SrcId nVis
0   C   1331+305000* 13:31:08.2873 +30.30.32.9590 J2000  0    4509
1   A   1445+099000* 14:45:16.4656 +09.58.36.0730 J2000  1    5292
2           N5921_2      15:22:00.0000 +05.04.00.0000 J2000  2   12852
      (nVis = Total number of time/baseline visibilities per field)
Spectral Windows: (1 unique spectral windows and 1 unique polarization setups)
SpwID #Chans Frame Ch1(MHz)  ChanWid(kHz) TotBW(kHz) Ref(MHz) Corrs
0      63 LSRK 1412.66507 24.4140625 1550.19688 1413.42801 RR LL
Sources: 3
ID  Name      SpwId RestFreq(MHz) SysVel(km/s)
0   1331+305000* 0    1420.405752 0
1   1445+099000* 0    1420.405752 0
2   N5921_2      0    1420.405752 0
Antennas: 27:
ID  Name  Station  Diam.  Long.  Lat.
0   1     VLA:N7   25.0 m -107.37.07.2 +33.54.12.9
1   2     VLA:W1   25.0 m -107.37.05.9 +33.54.00.5
2   3     VLA:W2   25.0 m -107.37.07.4 +33.54.00.9
3   4     VLA:E1   25.0 m -107.37.05.7 +33.53.59.2
4   5     VLA:E3   25.0 m -107.37.02.8 +33.54.00.5
5   6     VLA:E9   25.0 m -107.36.45.1 +33.53.53.6
```


6	7	VLA:E6	25.0 m	-107.36.55.6	+33.53.57.7
7	8	VLA:W8	25.0 m	-107.37.21.6	+33.53.53.0
8	9	VLA:N5	25.0 m	-107.37.06.7	+33.54.08.0
9	10	VLA:W3	25.0 m	-107.37.08.9	+33.54.00.1
10	11	VLA:N4	25.0 m	-107.37.06.5	+33.54.06.1
11	12	VLA:W5	25.0 m	-107.37.13.0	+33.53.57.8
12	13	VLA:N3	25.0 m	-107.37.06.3	+33.54.04.8
13	14	VLA:N1	25.0 m	-107.37.06.0	+33.54.01.8
14	15	VLA:N2	25.0 m	-107.37.06.2	+33.54.03.5
15	16	VLA:E7	25.0 m	-107.36.52.4	+33.53.56.5
16	17	VLA:E8	25.0 m	-107.36.48.9	+33.53.55.1
17	18	VLA:W4	25.0 m	-107.37.10.8	+33.53.59.1
18	19	VLA:E5	25.0 m	-107.36.58.4	+33.53.58.8
19	20	VLA:W9	25.0 m	-107.37.25.1	+33.53.51.0
20	21	VLA:W6	25.0 m	-107.37.15.6	+33.53.56.4
21	22	VLA:E4	25.0 m	-107.37.00.8	+33.53.59.7
23	24	VLA:E2	25.0 m	-107.37.04.4	+33.54.01.1
24	25	VLA:N6	25.0 m	-107.37.06.9	+33.54.10.3
25	26	VLA:N9	25.0 m	-107.37.07.8	+33.54.19.0
26	27	VLA:N8	25.0 m	-107.37.07.5	+33.54.15.8
27	28	VLA:W7	25.0 m	-107.37.18.4	+33.53.54.8

using the (default) `verbose=True` option. The most useful extra information that `verbose=True` gives is the list of the scans in the dataset.

2.2.6 Listing MS data (`listvis`)

The `listvis` task will print to the terminal (or file) listing of the data in your MS. The inputs are:

```
# listvis :: List measurement set visibilities.
vis          =      ''      # Name of input visibility file
options      =      'ap'    # List options: ap only
datacolumn  =      'data'   # Column to list: data, corrected, model, residual
field       =      ''      # Field names or index to be listed: ''==>all
spw         =      '*'     # Spectral window:channels: '\*'==>all, spw='1:5~57'
selectdata  =      False   # Other data selection parameters
average     =      ''      # Averaging mode: ==>none (Not yet implemented)
showflags   =      False   # Show flagged data (Not yet implemented)
pagerows    =      50     # Rows per page
listfile    =      ''      # Output file
async       =      False   # If true the taskname must be started using listvis(...)
```

For example:

```
CASA <2>: listvis('ngc5921.demo.ms',datacolumn='corrected',field='0',selectdata=T,antenna='*&*',pagerows=
All selected data has FIELD = 0
All selected data has SPW = 0
Date/Time:                RR:                LL:
```

```

1995/04/13/ Intrl UVDist Chn      Amp      Phs Wt F      Amp      Phs Wt F      U      V      W
-----|---|-----|---|-----|-----|-----|-----|-----|-----|
09:19:00.0 1-2  1823  0:  7.388  17.4 100    3.518 131.9 81    386.69 130409.94 15.46
09:19:00.0 1-2  1823  1: 11.913  -6.0 100   10.945  12.7 81    386.69 130409.94 15.46
09:19:00.0 1-2  1823  2: 13.570   1.5 100   12.700   3.5 81    386.69 130409.94 15.46
09:19:00.0 1-2  1823  3: 13.829  -0.6 100   13.479   2.2 81    386.69 130409.94 15.46
09:19:00.0 1-2  1823  4: 14.082   1.1 100   14.201  -0.1 81    386.69 130409.94 15.46
09:19:00.0 1-2  1823  5: 14.436   3.2 100   14.886   0.3 81    386.69 130409.94 15.46
09:19:00.0 1-2  1823  6: 15.190  -1.2 100   14.977  -1.0 81    386.69 130409.94 15.46
09:19:00.0 1-2  1823  7: 14.823   2.6 100   14.475   0.9 81    386.69 130409.94 15.46
09:19:00.0 1-2  1823  8: 14.484   1.7 100   14.572   0.9 81    386.69 130409.94 15.46
09:19:00.0 1-2  1823  9: 14.393   1.3 100   14.675   1.8 81    386.69 130409.94 15.46
Type Q to quit, A to toggle long/short list, or RETURN to continue [continue]: Q

```

ALERT: We are working on improving the format of the `listvis` output.

2.2.7 Listing and manipulating MS metadata (`vishead`)

The `vishead` task is provided to access keyword information in the Measurement Set. The default inputs are:

```

# vishead :: List, get, and put metadata in a measurement set
vis          =      ''      # Name of input visibility file
mode         =      'list'   # options: list, summary, get, put
  listitems  =      []      # items to list ([] for all)
async        =      False   #

```

The `mode = 'summary'` option just gives the same output as `listobs`.

For `mode = 'list'` the options are: `'telescope'`, `'observer'`, `'project'`, `'field'`, `'freq_group_name'`, `'spw_name'`, `'schedule'`, `'schedule_type'`, `'release_date'`.

```

CASA <29>: vishead('ngc5921.demo.ms',mode='list',listitems=[])
Out[29]:
{'cal_grp': (array([-1, -1, -1], dtype=int32), {}),
 'field': (array(['1331+30500002_0', '1445+09900002_0', 'N5921_2'],
  dtype='|S16'),
  {}),
 'fld_code': (array(['C', 'A', ''],
  dtype='|S2'), {}),
 'freq_group_name': (array(['none'],
  dtype='|S5'), {}),
 'log': ({'r1': False}, {}),
 'observer': (array(['TEST'],
  dtype='|S5'), {}),
 'project': (array([''],
  dtype='|S1'), {}),

```

```

'ptcs': ({'r1': array([[[-2.74392758]],
    [[ 0.53248521]]]),
    'r2': array([[[-2.42044692]],
    [[ 0.17412604]]]),
    'r3': array([[[-2.26020138]],
    [[ 0.08843002]]]),
    {'MEASINFO': {'Ref': 'J2000', 'type': 'direction'},
    'QuantumUnits': array(['rad', 'rad'],
    dtype='|S4')}}),
'release_date': (array([ 4.30444800e+09]),
    {'MEASINFO': {'Ref': 'TAI', 'type': 'epoch'},
    'QuantumUnits': array(['s'],
    dtype='|S2')}),
'schedule': ({'r1': False}, {}),
'schedule_type': (array(['']),
    dtype='|S1'), {}),
'source_name': (array(['1331+30500002_0', '1445+09900002_0', 'N5921_2'],
    dtype='|S16'),
    {}),
'spw_name': (array(['none'],
    dtype='|S5'), {}),
'telescope': (array(['VLA'],
    dtype='|S4'), {})}

```

You can use `mode='get'` to retrieve the values of specific keywords, and likewise `mode='put'` to change them. The inputs are:

```

mode          =      'get'      # options: list, summary, get, put
hdkey         =      ''         # keyword to get/put
hdindex       =      ''         # keyword index to get/put, counting from zero. ==>all

```

and

```

# vishead :: List, summary, get, and put metadata in a measurement set
mode          =      'put'      # options: list, summary, get, put
hdkey         =      ''         # keyword to get/put
hdindex       =      ''         # keyword index to get/put, counting from zero. ==>all
hdvalue       =      ''         # value of hdkey

```

For example, a common operation is to change the Telescope name (e.g. if it is unrecognized), e.g.

```

CASA <36>: vishead('ngc5921.demo.ms',mode='get',hdkey='telescope')
Out[36]:
(array(['VLA'],
    dtype='|S4'), {})

```

```
CASA <37>: vishead('ngc5921.demo.ms',mode='put',hdkey='telescope',hdvalue='EVLA')
```

```
CASA <38>: vishead('ngc5921.demo.ms',mode='get',hdkey='telescope')
```

```
Out [38]:
(array(['EVLA'],
      dtype='|S5'), {})
```

2.2.8 MS statistics (visstat)

ALERT: This is still a prototype task.

The `visstat` task is provided to obtain simple statistics for a Measurement Set, useful in regression tests.

The inputs are:

```
# visstat :: Displays statistical information from a measurement set
vis          =      ''      # Name of input visibility file
axis        =      'amp'    # Which values to use
  datacolumn =      'data'  # Which data column to use (data, corrected, model)

useflags    =      True    # Take flagging into account?
spw         =      ''      # spectral-window/frequency/channel
field       =      ''      # Field names or field index numbers: ''==>all, field='0~2,3C286'
selectdata  =      True    # More data selection parameters (antenna, timerange etc)
  antenna   =      ''      # antenna/baselines: ''==>all, antenna = '3,VA04'
  timerange =      ''      # time range: ''==>all, timerange='09:14:0~09:54:0'
  correlation =      ''    # Select data based on correlation
  scan      =      ''      # scan numbers: ''==>all
  array     =      ''      # (sub)array numbers: ''==>all
  uvrange   =      ''      # uv range: ''==>all; uvrange = '0~100klambda', default units=meters

async       =      False   # If true the taskname must be started using visstat(...)
```

Running this task returns a record (Python dictionary) with the statistics, which can be captured in a Python variable. For example,

```
CASA <42>: mystat = visstat('ngc5921.demo.ms',axis='amp',datacolumn='corrected',field='0')
```

```
CASA <43>: mystat
```

```
Out [43]:
{'CORRECTED': {'max': 51.938671112060547,
               'mean': 14.796444141750133,
               'medabsdevmed': 0.28020858764648438,
               'median': 14.764373779296875,
               'min': 0.81362706422805786,
               'npts': 514916.0,
               'quartile': 0.56053066253662109,
               'rms': 14.829294204711914,
```

```
'stddev': 0.98650836609147285,
'sum': 7618925.8316934109,
'sumsq': 113234125.12642419,
'var': 0.97319875636846753}}
```

```
CASA <44>: print mystat['CORRECTED']['stddev']
0.986508366091
```

The options for `axis` are:

```
axis='amplitude'      # or ('amp')
axis='phase'
axis='imag'
axis='scan_number'
axis='flag'
```

The phase of a complex number is in radians with range $(-\pi, \pi)$.

2.2.9 Concatenating multiple datasets (`concat`)

Once you have your data in the form of CASA Measurement Sets, you can go ahead and process your data using the editing, calibration, and imaging tasks. In some cases, you will most efficiently operate on single MS for a particular session (such as calibration). Other tasks will (eventually) take multiple Measurement Sets as input. For others, it is easiest to combine your multiple data files into one.

If you need to combine multiple datasets, you can use the `concat` task. The default inputs are:

```
# concat :: Concatenate two or more visibility data sets.
vis      =   [''] # Name of input visibility files to be concatenated
concatvis =   '' # Name of output visibility file
freqtol  =   '' # Frequency tolerance for considering data as the same spwid
dirtol   =   '' # Direction tolerance for considering data as the same field
timesort = False # If true, sort by TIME in ascending order
async    = False # If true the taskname must be started using concat(...)
```

The `vis` parameter will take a list of one or more MS. Usually, this will contain all the MS to combine.

The `concatvis` parameter contains the name of the output MS. If this points to an existing file on disk, then the MS in `vis` will be appended to it, otherwise a new MS file is created to contain the concatenated data. Be careful here!

The `timesort` parameter can be used to make sure the output MS is in time order (e.g. if your input MS have concurrent times). This can possibly speed up some subsequent calibration operations.

The parameters `freqtol` and `dirtol` control how close together in frequency and angle on the sky spectral windows or field locations need to be before calling them the same.

ALERT: Note that if multiple frequencies or pointings are combined using `frequ_tol` or `dirtol`, then the data are not changed (ie. not rephased to the single phase center). Use of these parameters is intended to be tolerant of small offsets (e.g. planets tracked which move slightly in J2000 over the course of observations, or combining epochs observed with slightly different positions).

For example:

```
default('concat')
vis = ['n4826_16apr.split.ms', 'n4826_22apr.split.ms']
concatvis = 'n4826_tboth.ms'
frequ_tol = '50MHz'
concat()
```

combines the two days in `'n4826_16apr.split.ms'` and `'n4826_22apr.split.ms'` into a new output MS called `'n4826_tboth.ms'`.

ALERT: There are additional issues when concatenating individual MS files with different spectral windows. In this case, the rest frequencies (if set) of these windows may be lost in subsequent operations (e.g. `split`). If this occurs, then you can either set the `restfreq` explicitly in certain tasks (`plotxy` and `clean`), or use the following snippet of code to fix this up after `concat` has run:

```
#
#####
#
# Fix up the MS after concat (NOTE: STILL NECESSARY IN 2.4 and 3.0)
# This ensures that the rest freq will be found for all spws.
# print '--Fixing up spw rest frequencies in MS--'
vis='ngc4826.tutorial.ms'          # Example vis name, put yours here!
tb.open(vis+'/SOURCE',nomodify=false)
spwid=tb.getcol('SPECTRAL_WINDOW_ID')
# The following is for 64bit systems 08-Jul-2008. ok on 32bit also.
spwid.setfield(-1,'int32')
tb.putcol('SPECTRAL_WINDOW_ID',spwid)
tb.close()
```

For example, this is used in the NGC4826 BIMA mosaic script (Appendix F.3).

2.3 Data Selection

Once in MS form, subsets of the data can be operated on using the tasks and tools. In CASA, there are three common data selection parameters used in the various tasks: `field`, `spw`, and `selectdata`. In addition, the `selectdata` parameter, if set to `True`, will open up a number of other sub-parameters for selection. The selection operation is unified across all the tasks. The available `selectdata` parameters may not be the same in all tasks. But if present, the same parameters mean the same thing and behave in the same manner when used in any task.

For example:

```

field          =          ''          # field names or index of calibrators ''==>all
spw            =          ''          # spectral window:channels: ''==>all
selectdata    =          False       # Other data selection parameters

versus

field          =          ''          # field names or index of calibrators ''==>all
spw            =          ''          # spectral window:channels: ''==>all
selectdata    =          True        # Other data selection parameters
  timerange   =          ''          # time range: ''==>all
  uvrange     =          ''          # uv range''=all
  antenna     =          ''          # antenna/baselines: ''==>all
  scan        =          ''          # scan numbers: Not yet implemented
  msselect    =          ''          # Optional data selection (Specialized. but see help)

```

The following are the general syntax rules and descriptions of the individual selection parameters of particular interest for the tasks:

2.3.1 General selection syntax

Most of the selections are effected through the use of selection strings. This sub-section describes the general rules used in constructing and parsing these strings. Note that some selections are done though the use of numbers or lists. There are also parameter-specific rules that are described under each parameter.

All lists of basic selection specification-units are comma separated lists and can be of any length. White-spaces before and after the commas (e.g. '3C286, 3C48, 3C84') are ignored, while white-space within sub-strings is treated as part of the sub-string (e.g. '3C286, VIRGO A, 3C84').

All integers can be of any length (in terms of characters) composed of the characters 0–9. Floating point numbers can be in the standard format (DIGIT.DIGIT, DIGIT., or .DIGIT) or in the mantissa-exponent format (e.g. 1.4e9). Places where only integers make sense (e.g. IDs), if a floating point number is given, only the integer part is used (it is truncated).

Range of numbers (integers or real numbers) can be given in the format 'N0~N1'. For integer ranges, it is expanded into a list of integers starting from N0 (inclusive) to N1 (inclusive). For real numbers, it is used to select all values present for the appropriate parameter in the Measurement Set between N0 and N1 (including the boundaries). Note that the '~' character is used rather than the more obvious '-' in order to accommodate hyphens in strings and minus signs in numbers.

Wherever appropriate, units can be specified. The units are used to convert the values given to the units used in the Measurement Set. For ranges, the unit is specified only once (at the end) and applies to both the range boundaries.

2.3.1.1 String Matching

String matching can be done in three ways. Any component of a comma separated list that cannot be parsed as a number, a number range, or a physical quantity is treated as a regular expression or a literal string. If the string does not contain the characters '*', '{', '}' or '?', it is treated as a literal string and used for exact matching. If any of the above mentioned characters are part of the string, they are used as a regular expression. As a result, for most cases, the user does not need to supply any special delimiters for literal strings and/or regular expressions. For example:

```
field = '3'      # match field ID 3 and not select field named "3C286".

field = '3*'    # used as a pattern and matched against field names. If
                # names like "3C84", "3C286", "3020+2207" are found,
                # all will match. Field ID 3 will not be selected
                # (unless of course one of the above mentioned field
                # names also correspond to field ID 3!).

field = '30*'   # will match only with "3020+2207" in above set.
```

However if it is required that the string be matched exclusively as a regular expression, it can be supplied within a pair of '/' as delimiters (e.g. '/.+BAND.+/'). A string enclosed within double quotes ('"') is used exclusively for pattern matching (patterns are a simplified form of regular expressions - used in most UNIX commands for string matching). Patterns are internally converted to equivalent regular expressions before matching. See the Unix command "info regex", or visit <http://www.regular-expressions.info>, for details of regular expressions and patterns.

Strings can include any character except the following:

```
','  ';'  '"'  '/'  NEWLINE
```

(since these are part of the selection syntax). Strings that do not contain any of the characters used to construct regular expressions or patterns are used for exact matches. Although it is highly discouraged to have name in the MS containing the above mentioned reserved characters, if one *does* choose to include the reserved characters as parts of names etc., those names can only be matched against quoted strings (since regular expression and patterns are a super-set of literal strings - i.e., a literal string is also a valid regular expression).

This leaves '"', '*', '{', '}' or '?' as the list of printable character that cannot be part of a name (i.e., a name containing this character can never be matched in a MSSelection expression). These will be treated as pattern-matching even inside double double quotes ('" " "). There is currently no escape mechanism (e.g. via a backslash).

Some examples of strings, regular expressions, and patterns:

- The string 'LBAND' will be used as a literal string for exact match. It will match only the exact string LBAND.
- The wildcarded string '*BAND*' will be used as a string pattern for matching. This will match any string which has the sub-string BAND in it.

- The string `"*BAND*"` will also be used as a string pattern, matching any string which has the sub-string `BAND` in it.
- The string `"/.+BAND.+/"` will be used as a regular expression. This will also match any string which has the sub-string `BAND` in it. (the `.+` regex operator has the same meaning as the `*` wildcard operator of patterns).

2.3.2 The field Parameter

The `field` parameter is a string that specifies which field names or ids will be processed in the task or tool. The field selection expression consists of comma separated list of field specifications inside the string.

Field specifications can be literal field names, regular expressions or patterns (see § 2.3.1.1). Those fields for which the entry in the `NAME` column of the `FIELD MS` sub-table match the literal field name/regular expression/pattern are selected. If a field name/regular expression/pattern fails to match any field name, the given name/regular expression/pattern are matched against the field code. If still no field is selected, an exception is thrown.

Field specifications can also be given by their integer IDs. IDs can be a single or a range of IDs. Field ID selection can also be done as a boolean expression. For a field specification of the form `'>ID'`, all field IDs greater than `ID` are selected. Similarly for `'<ID'` all field IDs less than the `ID` are selected.

For example, if the `MS` has the following observations:

MS summary:

=====

FIELDID	SPWID	NChan	Pol	NRows	Source Name
0	0	127	RR	10260	0530+135
1	0	127	RR	779139	05582+16320
2	0	127	RR	296190	05309+13319
3	0	127	RR	58266	0319+415
4	0	127	RR	32994	1331+305
5	1	1	RR,RL,LL,RR	23166	KTIP

one might select

```
field = '0~2,KTIP'           # FIELDID 0,1,2 and field name KTIP
field = '0530+135'          # field 0530+135
field = '05*'                # fields 0530+135,05582+16320,05309+13319
```

2.3.3 The spw Parameter

The `spw` parameter is a string that indicates the specific spectral windows and the channels within them to be used in subsequent processing. Spectral window selection (`'SPWSEL'`) can be given as

a spectral window integer ID, a list of integer IDs, a spectral window name specified as a literal string (for exact match) or a regular expression or pattern.

The specification can be via frequency ranges or by indexes. A range of frequencies are used to select all spectral windows which contain channels within the given range. Frequencies can be specified with an optional unit — the default unit being Hz. Other common choices for radio and mm/sub-mm data are kHz, MHz, and GHz. You will get the entire spectral windows, not just the channels in the specified range. You will need to do channel selection (see below) to do that.

The `spw` can also be selected via comparison for integer IDs. For example, `>ID` will select all spectral windows with ID greater than the specified value, while `<ID` will select those with ID lesser than the specified value.

ALERT: In the current release, `<ID` and `>ID` are *inclusive* with the ID specified included in the selection, e.g. `spw=<2` is equivalent to `spw='0,1,2'` and not `spw='0,1'` as was intended. This will be fixed in an upcoming release.

Spectral window selection using strings follows the standard rules:

```
spw = '1'           # SPWID 1
spw = '1,3,5'      # SPWID 1,3,5
spw = '0~3'        # SPWID 0,1,2,3
spw = '0~3,5'      # SPWID 0,1,2,3 and 5
spw = '<3,5'        # SPWID 0,1,2,3 and 5
spw = '*'          # All spectral windows
spw = '1412~1415MHz' # Spectral windows containing 1412-1415MHz
```

In some cases, the spectral windows may allow specification by name. For example,

```
spw = '3mmUSB, 3mmLSB' # choose by names (if available)
```

might be meaningful for the dataset in question.

Note that the order in which multiple `spws` are given may be important for other parameters. For example, the `mode = 'channel'` in `clean` uses the first `spw` as the origin for the channelization of the resulting image cube.

2.3.3.1 Channel selection in the `spw` parameter

Channel selection can be included in the `spw` string in the form `'SPWSEL:CHANSEL'` where `CHANSEL` is the channel selector. In the end, the spectral selection within a given spectral window comes down to the selection of specific channels. We provide a number of shorthand selection options for this. These `CHANSEL` options include:

- *Channel ranges:* `'START~STOP'`

ALERT!

Not all options are available yet, such as percentages or velocities. Stay tuned!

- *Frequency ranges:* 'FSTART~FSTOP'
- *Velocity ranges:* 'VSTART~VSTOP' (**not yet available**)
- *Bandwidth percentages:* 'PSTART~PSTOP' or 'PWIDTH' (**not yet available**)
- *Channel striding/stepping:* 'START~STOP^STEP' or 'FSTART~FSTOP^FSTEP'

The most common selection is via channel ranges 'START~STOP' or frequency ranges 'FSTART~FSTOP':

```
spw = '0:13~53'           # spw 0, channels 13-53, inclusive
spw = '0:1413~1414MHz'   # spw 0, 1413-1414MHz section only
```

All ranges are inclusive, with the channel given by, or containing the frequency or velocity given by, **START** and **STOP** plus all channels between included in the selection. You can also select the spectral window via frequency ranges 'FSTART~FSTOP', as described above:

```
spw = '1413~1414MHz:1413~1414MHz' # channels falling within 1413~1414MHz
spw = '*:1413~1414MHz'           # does the same thing
```

You can also specify multiple spectral window or channel ranges, e.g.

```
spw = '2:16, 3:32~34'          # spw 2, channel 16 plus spw 3 channels 32-34
spw = '2:1~3;57~63'           # spw 2, channels 1-3 and 57-63
spw = '1~3:10~20'             # spw 1-3, channels 10-20
spw = '*:4~56'                # all spw, channels 4-56
```

Note the use of the wildcard in the last example.

A step can be also be included using '^STEP' as a postfix:

```
spw = '0:10~100^2'           # chans 10,12,14,...,100 of spw 0
spw = ':^4'                  # chans 0,4,8,... of all spw
spw = ':100~150GHz^10GHz'    # closest chans to 100,110,...,150GHz
```

A step in frequency or velocity will pick the channel in which that frequency or velocity falls, or the nearest channel.

2.3.4 The selectdata Parameters

The `selectdata` parameter, if set to `True`, will expand the inputs to include a number of sub-parameters, given below and in the individual task descriptions (if different). If `selectdata = False`, then the sub-parameters are treated as blank for selection by the task. The default for `selectdata` is `False`.

The common `selectdata` expanded sub-parameters are:

2.3.4.1 The antenna Parameter

The `antenna` selection string is a semi-colon (';') separated list of baseline specifications. A baseline specification is of the form:

- `'ANT1'` — Select all baselines including the antenna(s) specified by the selector `ANT1`.
- `'ANT1&'` — Select only baselines between the antennas specified by the selector `ANT1`.
- `'ANT1&ANT2'` — Select only the cross-correlation baselines between the antennas specified by selector `ANT1` and antennas specified by selector `ANT2`. Thus `'ANT1&'` is an abbreviation for `'ANT1&ANT1'`.
- `'ANT1&&ANT2'` — Select only auto-correlation and cross-correlation baselines between antennas specified by the selectors `ANT1` and `ANT2`. Note that this is what the default `antenna=''` gives you.
- `'ANT1&&&'` — Select only autocorrelations specified by the selector `ANT1`.

The selectors `ANT1` and `ANT2` are comma-separated lists of antenna integer-IDs or literal antenna names, patterns, or regular expressions. The `ANT` strings are parsed and converted to a list of antenna integer-IDs or IDs of antennas whose name match the given names/pattern/regular expression. Baselines corresponding to all combinations of the elements in lists on either side of ampersand are selected.

Integer IDs can be specified as single values or a range of integers. When items of the list are parsed as literal strings or regular expressions or patterns (see § 2.3.1 for more details on strings). All antenna names that match the given string (exact match)/regular expression/pattern are selected.

ALERT: Just for antenna selection, a user supplied integer (or integer list) is converted to a string and matched against the antenna name. If that fails, the normal logic of using an integer as an integer and matching it with antenna index is done. Note that currently there is no method for specifying a pure index (e.g. a number that will not first be checked against the name).

The comma is used only as a separator for the list of antenna specifications. The list of baselines specifications is a semi-colon separated list, e.g.

```
antenna = '1~3 & 4~6 ; 10&11'
```

will select baselines between antennas 1,2,3 and 4,5,6 (`'1&4'`, `'1&5'`, ..., `'3&6'`) plus baseline `'10&11'`.

The wildcard operator (`'*'`) will be the most often used pattern. To make it easy to use, the wildcard (and only this operator) can be used without enclosing it in quotes. For example, the selection

```
antenna = 'VA*'
```

will match all antenna names which have 'VA' as the first 2 characters in the name (irrespective of what follows after these characters).

There is also a negation operator "!" that can be used to de-select antennas or baselines.

Some examples:

```

antenna=' '          # shows blank autocorr pages
antenna='*&*'       # does not show the autocorrs
antenna='*&&*'      # show both auto and cross-cor (default)
antenna='*&&&*'     # shows only autocorrs

antenna='5&*'       # shows non-auto baselines with AN 5

antenna='5,6&&&'    # AN 5 and 6 autocor
antenna='5&&&;6&*'  # AN 5 autocor plus cross-cors to AN 6

antenna='!5'        # baselines not involving AN 5

```

Antenna numbers as names: Needless to say, naming antennas such that the names can also be parsed as a valid token of the syntax is a bad idea. Nevertheless, antenna names that contain any of the reserved characters and/or can be parsed as integers or integer ranges can still be used by enclosing the antenna names in double quotes (' "ANT" '). E.g. the string

```
antenna = '10~15,21,VA22'
```

will expand into an antenna ID list 10,11,12,13,14,15,21,22 (assuming the index of the antenna named 'VA22' is 22). If, however, the antenna with ID index 50 is named '21', then the string

```
antenna = '10~15,21,VA22'
```

will expand into an antenna ID list of 10,11,12,13,14,15,50,22. *Keep in mind that numbers are FIRST matched against names, and only against indices if that matching fails.* There is currently no way to force a selection to use the index, and if there an antenna with that name it will select that.

Read elsewhere (e.g. `info regex` under Unix) for details of regular expression and patterns.

2.3.4.2 The scan Parameter

The `scan` parameter selects the scan ID numbers of the data. There is currently no naming convention for scans. The scan ID is filled into the MS depending on how the data was obtained, so use this with care.

Examples:

```

scan = '3'           # scan number 3.
scan = '1~8'        # scan numbers 1 through 8, inclusive
scan = '1,2,4,6'    # scans 1,2,4,6
scan = '<9'          # scans <9 (1-8)

```

NOTE: ALMA and VLA/EVLA number scans starting with 1 and not 0. You can see what the numbering is in your MS using the `listobs` task with `verbose=True` (see § 2.2.5).

2.3.4.3 The `timerange` Parameter

The time strings in the following (`T0`, `T1` and `dT`) can be specified as `YYYY/MM/DD/HH:MM:SS.FF`. The time fields (i.e., `YYYY`, `MM`, `DD`, `HH`, `MM`, `SS` and `FF`), starting from left to right, may be omitted and they will be replaced by context sensitive defaults as explained below.

Some examples:

1. `timerange='T0~T1'`: Select all time stamps from `T0` to `T1`. For example:

```
timerange = '2007/10/09/00:40:00 ~ 2007/10/09/03:30:00'
```

Note that fields missing in `T0` are replaced by the fields in the time stamp of the first valid row in the MS. For example,

```
timerange = '09/00:40:00 ~ 09/03:30:00'
```

where the `YY/MM/` part of the selection has been defaulted to the start of the MS.

Fields missing in `T1`, such as the date part of the string, are replaced by the corresponding fields of `T0` (after its defaults are set). For example:

```
timerange = '2007/10/09/22:40:00 ~ 03:30:00'
```

does the same thing as above.

2. `timerange='T0'`: Select all time stamps that are within an integration time of `T0`. For example,

```
timerange = '2007/10/09/23:41:00'
```

Integration time is determined from the first valid row (more rigorously, an average integration time should be computed). Default settings for the missing fields of `T0` are as in (1).

3. `timerange='T0+dT'`: Select all time stamps starting from `T0` and ending with time stamp `T0+dT`. For example,

```
timerange = '23:41:00+01:00:00'
```

picks an hour-long chunk of time.

Defaults of `T0` are set as usual. Defaults for `dT` are set from the time corresponding to `MJD=0`. Thus, `dT` is a specification of length of time from the assumed nominal "start of time".

4. `timerange='>T0'`: Select all times greater than `T0`. For example,

```
timerange = '>2007/10/09/23:41:00'
timerange = '>23:41:00'           # Same thing without day specification
```

Default settings for T0 are as above.

5. `timerange='<T1'`: Select all times less than T1. For example,

```
timerange = '<2007/10/09/23:41:00'
```

Default settings for T1 are as above.

An ultra-conservative selection might be:

```
timerange = '1960/01/01/00:00:00~2020/12/31/23:59:59'
```

which would choose all possible data!

2.3.4.4 The `uvrange` Parameter

Rows in the MS can also be selected based on the uv-distance or physical baseline length that the visibilities in each row correspond to. This `uvrange` can be specified in various formats.

The basic building block of uv-distance specification is a valid number with optional units in the format `N[UNIT]` (the unit in square brackets is optional). We refer to this basic building block as `UVDIST`. The default unit is meter. Units of length (such as `'m'` and `'km'`) select physical baseline distances (independent of wavelength). The other allowed units are in wavelengths (such as `'lambda'`, `'klambda'` and `'Mlambda'` and are true uv-plane radii

$$r_{uv} = \sqrt{u^2 + v^2}. \quad (2.1)$$

If only a single `UVDIST` is specified, all rows, the uv-distance of which exactly matches the given `UVDIST`, are selected.

`UVDIST` can be specified as a range in the format `'N0~N1[UNIT]'` (where `N0` and `N1` are valid numbers). All rows corresponding to uv-distance between `N0` and `N1` (inclusive) when converted the specified units are selected.

`UVDIST` can also be selected via comparison operators. When specified in the format `'>UVDIST'`, all visibilities with uv-distances greater than the given `UVDIST` are selected. Likewise, when specified in the format `'<UVDIST'`, all rows with uv-distances less than the given `UVDIST` are selected.

Any number of above mentioned uv-distance specifications can be given as a comma-separated list.

Examples:

```
uvrange = '100~200km'           # an annulus in physical baseline length
uvrange = '24~35Mlambda, 40~45Mlambda' # two annuli in units of mega-wavelengths
uvrange = '< 45klambda'         # less than 45 kilolambda
uvrange = '> 0lambda'           # greater than zero length (no auto-corrs)
uvrange = '100km'              # baselines of length 100km
uvrange = '100klambda'         # uv-radius 100 kilolambda
```

2.3.4.5 The `mselect` Parameter

More complicated selections within the MS structure are possible using the Table Query Language (TaQL). This is accessed through the `mselect` parameter.

Note that the TaQL syntax does not follow the rules given in § 2.3.1 for our other selection strings. TaQL is explained in more detail in **Aips++ NOTE 199 — Table Query Language** (<http://aips2.nrao.edu/docs/notes/199/199.html>). This will eventually become a CASA document. The specific columns of the MS are given in the most recent MS specification document: **Aips++ NOTE 229 — MeasurementSet definition version 2.0** (<http://aips2.nrao.edu/docs/notes/229/229.html>). This documentation will eventually be updated to the CASA document system.

Most selection can be carried out using the other selection parameters. However, these are merely shortcuts to the underlying TaQL selection. For example, field and spectral window selection can be done using `mselect` rather than through `field` or `spw`:

```
mselect='FIELD_ID == 0'           # Field id 0 only
mselect='FIELD_ID <= 1'         # Field id 0 and 1
mselect='FIELD_ID IN [1,2]'     # Field id 1 and 2
mselect='FIELD_ID==0 && DATA_DESC_ID==3' # Field id 0 in spw id 3 only
```

ALERT: The `mselect` style parameters will be phased out of the tasks. TaQL selection will still be available in the Toolkit.

Chapter 3

Data Examination and Editing

3.1 Plotting and Flagging Visibility Data in CASA

The tasks available for plotting and flagging of data are:

- `flagmanager` — manage versions of data flags (§ 3.2)
- `flagautocorr` — non-interactive flagging of auto-correlations (§ 3.3)
- `plotms` — create X-Y plots of data in MS, flag data (§ 3.4.1)
- `plotxy` — older X-Y plotter with some functionalities not yet implemented in `plotms` (§ 3.4.2)
- `flagdata` --- non-interactive flagging of data (§ 3.5)
- `browsetable` --- browse data in any CASA table (including a MS) (§ 3.6)
- `plotants` --- create simple plots of antenna positions (§ 3.4.3)

The following sections describe the use of these tasks.

Information on other related operations can be found in:

- `listobs` — list summary of a MS (§ 2.2.5)
- `listvis` — list data in a MS (§ 2.2.6)
- `selectdata` — general data selection syntax (§ 2.3)
- `viewer` — use the `casaviewer` to display the MS as a raster image, and flag it (§ 7)

3.2 Managing flag versions with flagmanager

The `flagmanager` task will allow you to manage different versions of flags in your data. These are stored inside a CASA `flagversions` table, under the name of the MS `<msname>.flagversions`. For example, for the MS `jupiter6cm.usecase.ms`, there will need to be `jupiter6cm.usecase.ms.flagversions` on disk. This is created on import (by `importvla` or `importuvfits`) or when flagging is first done on an MS without a `.flagversions` (e.g. with `plotxy`).

By default, when the `.flagversions` is created, this directory will contain a `flags.Original` in it containing a copy of the original flags in the MAIN table of the MS so that you have a backup. It will also contain a file called `FLAG.VERSION.LIST` that has the information on the various flag versions there. The `flagversions` are cumulative, ie. a specific version number contains all the flags from the lower version numbers, too.

The inputs for `flagmanager` are:

```
vis          =          ''          # Name of input visibility file (MS)
mode         =          'list'      # Flag management operation (list,save,restore,delete)
```

The `mode='list'` option will list the available flagversions from the `<msname>.flagversions` file. For example:

```
CASA <102>: default('flagmanager')
CASA <103>: vis = 'jupiter6cm.usecase.ms'
CASA <104>: mode = 'list'
CASA <105>: flagmanager()
MS : /home/imager-b/smyers/Oct07/jupiter6cm.usecase.ms
```

```
main : working copy in main table
Original : Original flags at import into CASA
flagautocorr : flagged autocorr
xyflags : Plotxy flags
```

The `mode` parameter expands the options. For example, if you wish to save the current flagging state of `vis=<msname>`,

```
mode          =          'save'      # Flag management operation (list,save,restore,delete)
  versionname =          ''          # Name of flag version (no spaces)
  comment     =          ''          # Short description of flag version
  merge       =          'replace'   # Merge option (replace, and, or)
```

with the output version name specified by `versionname`. For example, the above `xyflags` version was written using:

```
default('flagmanager')
vis = 'jupiter6cm.usecase.ms'
mode = 'save'
versionname = 'xyflags'
comment = 'Plotxy flags'
flagmanager()
```

and you can see that there is now a sub-table in the flagversions directory

```
CASA <106>: ls jupiter6cm.usecase.ms.flagversions/
IPython system call: ls -F jupiter6cm.usecase.ms.flagversions/
flags.flagautocorr flags.Original flags.xyflags FLAG_VERSION_LIST
```

It is recommended that you use this facility regularly to save versions during flagging.

You can restore a previously saved set of flags using the `mode='restore'` option:

```
mode          = 'restore'      # Flag management operation (list,save,restore,delete)
versionname   = ''            # Name of flag version (no spaces)
merge         = 'replace'     # Merge option (replace, and, or)
```

The `merge` sub-parameter will control how the flags are restored. For `merge='replace'`, the flags in `versionname` will replace those in the MAIN table of the MS. For `merge='and'`, only data that is flagged in BOTH the current MAIN table and in `versionname` will be flagged. For `merge='or'`, data flagged in EITHER the MAIN or in `versionname` will be flagged.

The `mode='delete'` option can be used to remove `versionname` from the flagversions:

```
mode          = 'delete'      # Flag management operation (list,save,restore,delete)
versionname   = ''            # Name of flag version (no spaces)
```

3.3 Flagging auto-correlations with flagautocorr

The `flagautocorr` task can be used if all you want to do is to flag the auto-correlations out of the MS. Nominally, this can be done upon filling from the VLA for example, but you may be working from a dataset that still has them.

This task has a single input, the MS file name:

```
vis           = ''           # Name of input visibility file (MS)
```

To use it, just set and go:

```
CASA <90>: vis = 'jupiter6cm.usecase.ms'
CASA <91>: flagautocorr()
```

Note that the auto-correlations can also be flagged using `flagdata` (§ 3.5) but the `flagautocorr` task is a handy shortcut for this common operation.

3.4 X-Y Plotting and Editing of the Data

There are three main X-Y plotting tasks in CASA:

- `plotms` — create X-Y plots of data in MS, flag data (§ 3.4.1)
- `plotxy` — older X-Y plotter with some functionalities not yet implemented in `plotms` (§ 3.4.2)
- `plotants` — create simple plots of antenna positions (§ 3.4.3)

3.4.1 MS Plotting and Editing using `plotms`

The principal way to get X-Y plots of visibility data is the `plotms` task. This task also provides editing capability. `Plotms` is a GUI-style plotter, based on Qt (<http://www.trolltech.com/qt>). It can either be started as a task within CASA (`plotms`) or from outside CASA (type `casaplotms` on the command line).

The current inputs to the `plotms` task are:

```
# plotms :: A plotter/interactive flagger for visibility data.
vis          =          ''      # input visibility dataset (blank for none)
xaxis        =          ''      # plot x-axis (blank for default/current)
yaxis        =          ''      # plot y-axis (blank for default/current)
selectdata   =      False      # data selection parameters
averagedata  =      False      # data averaging parameters
transform    =      False      # transform data in various ways?
extendflag   =      False      # have flagging extend to other data points?
async        =      False      # If true the taskname must be started using
                                # plotms(...)
```

All of these parameters can also be set or modified from inside the `plotms` window. Note that, if the `vis` parameter is set to the name of a measurement set here, when you start up `plotms`, the *entire* measurement set will be plotted, which can be time consuming. It is probably best to leave all parameters blank for now, setting them as needed inside the `plotms` GUI.

3.4.1.1 Loading and Selecting Data

When `plotms` is first started, a window will appear as in Figure 3.1. It will, by default, display the **Plots** tab (as chosen from the tabs at the top of the `plotms` window—e.g., **Plots**, **Flagging**, **Tools**...) and the **Plots > Data** tab (as chosen from the tabs on the far left side of the `plotms` window—e.g., **Data**, **Axes**, **Trans**...). First, a measurement set should be loaded by clicking on **Browse** near the top of the **Plots > Data** tab, and selecting a `.ms` directory (just select the directory itself; do not descend into the `.ms` directory). A plot can now be made of the measurement set by clicking on **Plot**—but beware, this would plot the entire measurement set, and could take quite some time! It is probably better to select a subset of the measurement set using the **Selection** windows in the **Plots > Data** tab before clicking **Plot**.

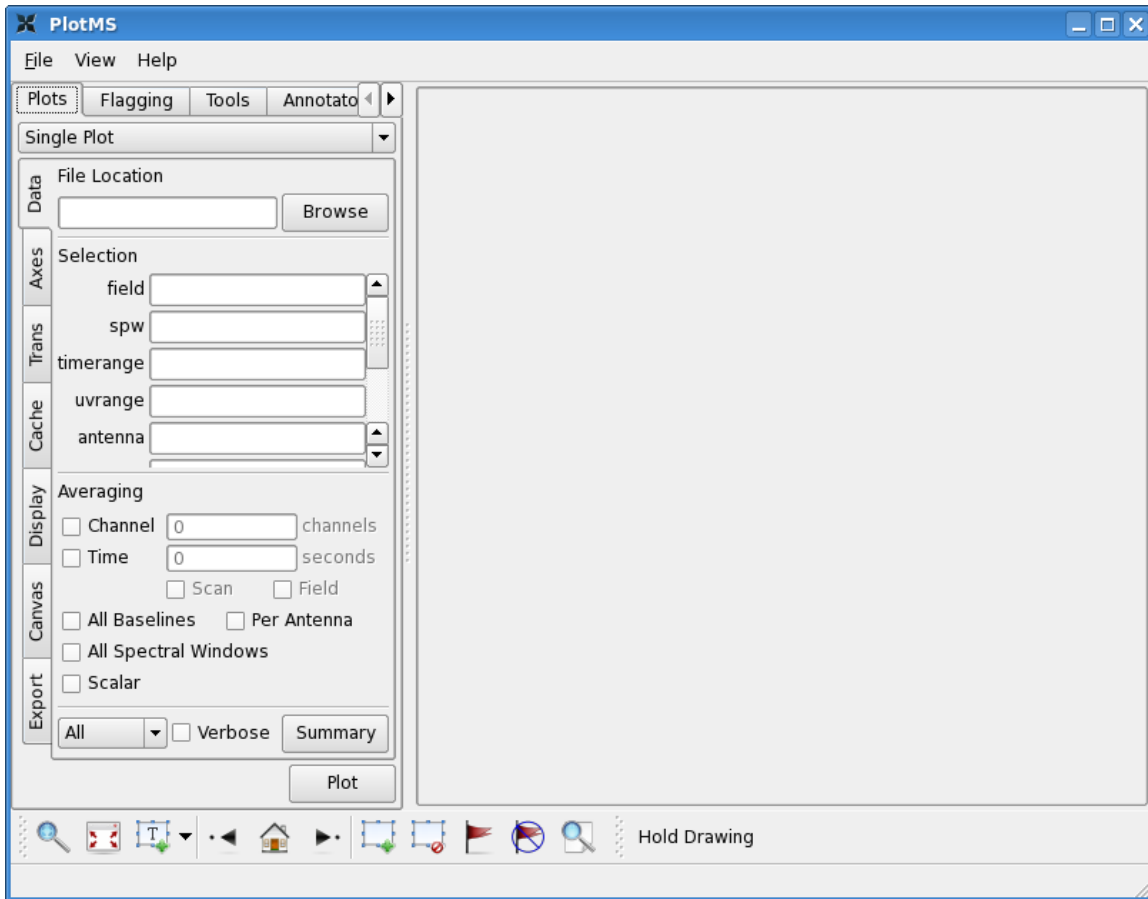


Figure 3.1: A freshly-started `plotms` GUI window. Note that the **Plots** > **Data** tab is selected, which is discussed in § 3.4.1.1, 3.4.1.5, and 3.4.1.7.

The options for data selection are:

- **field**
- **spw**
- **timerange**
- **uvrange**
- **antenna**
- **scan**
- **corr**
- **array**

- **msselect**

These are described in § 2.3. Note that, unlike when setting data selection parameters from the CASA command line, no quotation marks are needed around strings.

When a plotting parameter has been changed, it will turn red (for example, when a new measurement set is loaded, **File Location** turns red). This alerts the user that, if the **Plot** button is clicked, a change will be made to the displayed plot.

Once you have selected the desired subset of data, if you click **Plot**, **plotms** will by default plot amplitude versus time. See the next section for information about other possible axes.

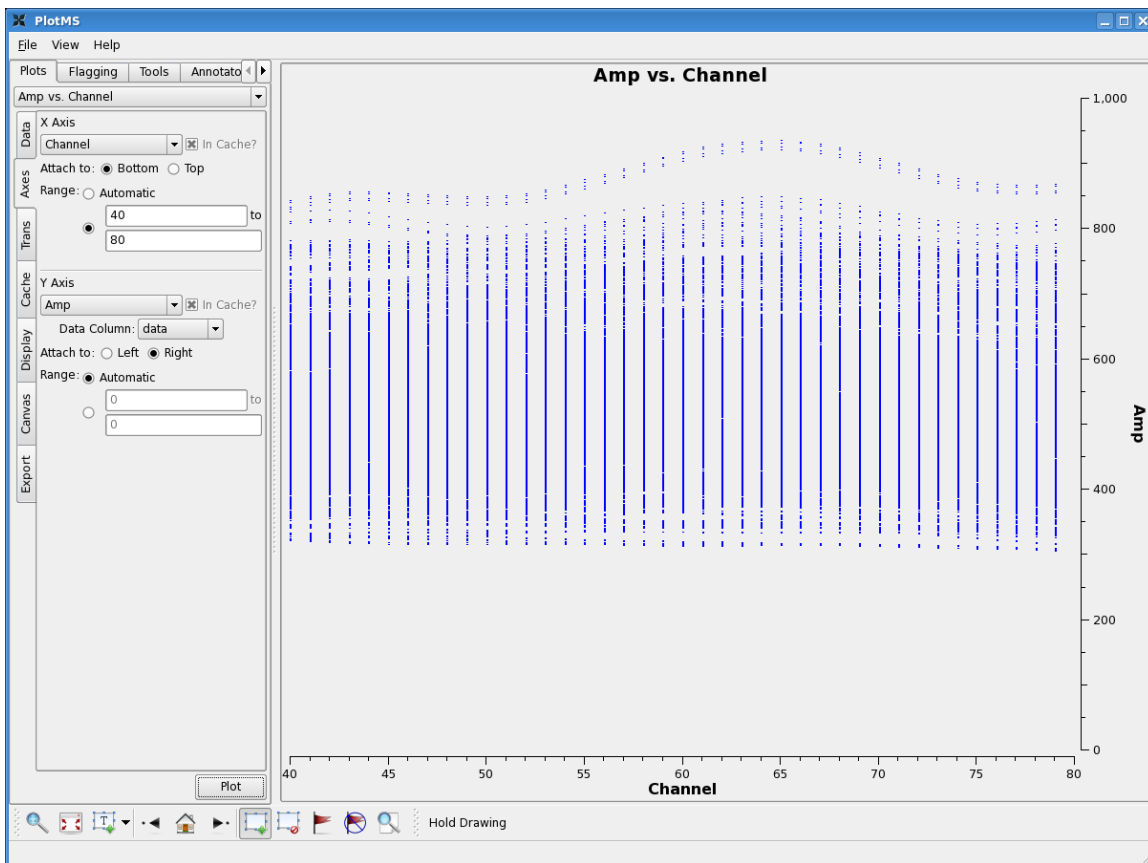


Figure 3.2: The **Plots > Axes** tab in the **plotms** GUI window, used to make a plot of **Amp** versus **Channel**.

3.4.1.2 Plot Axes

The X and Y axes of a plot are selected by clicking on the **Plots > Axes** tab on the left side of the **plotms** window, and choosing an entry from the drop-down menus below **X Axis** and **Y Axis**

(see Figure 3.2). Possible axes are:

- **Scan** — The scan number, as listed by `listobs` (§ 2.2.5) or the data summary in `plotms` (§ 3.4.1.7).
- **Field** — The field number, as listed by `listobs` (§ 2.2.5) or the `plotms` data summary (§ 3.4.1.7).
- **Time** — The time at which the visibility was observed, given in terms of calendar year (yyyy/mm/dd/hh:mm:ss.ss).
- **Time_interval** — The integration time in seconds.
- **Spw** — The spectral window number. The characteristics of each spectral window are listed in `listobs` (§ 2.2.5) or the `plotms` data summary (§ 3.4.1.7).
- **Channel** — The spectral channel number.
- **Frequency** — Frequency in units of GHz. The frame for the frequency (e.g., topocentric, barycentric, LSRK) can be set in the **Plots** > **Trans** tab (§ 3.4.1.8).
- **Velocity** — Velocity in units of km s^{-1} , as defined by the **Frame**, **Velocity Defn**, and **Rest Freq** parameters in the **Plots** > **Trans** tab (§ 3.4.1.8).
- **Corr** — Correlations which have been assigned integer IDs: 5 = RR; 6 = RL; 7 = LR; and 8 = LL.
- **Antenna1** — The first antenna in a baseline pair; for example, for baseline 2-4, `Antenna1` = 2. Antennae are numbered according to the antenna IDs listed in `listobs` (§ 2.2.5) or the `plotms` data summary (§ 3.4.1.7).
- **Antenna2** — The second antenna in a baseline pair; for baseline 2-4, `Antenna2` = 4. Antennae are numbered according to the antenna IDs listed in `listobs` (§ 2.2.5) or the `plotms` data summary (§ 3.4.1.7).
- **Antenna** — Antenna ID for plotting antenna-based quantities. Antennae are numbered according to the antenna IDs listed in `listobs` (§ 2.2.5) or the `plotms` data summary (§ 3.4.1.7).
- **Baseline** — The baseline number.
- **UVDist** — Projected baseline separations in units of meters. Note that **UVDist** is *not* a function of frequency.
- **UVDist_L** — Projected baseline separations in units of the observing wavelength (λ , not kilolambda). **UVDist_L** is a function of frequency, and therefore, there will be a different data point for each frequency channel.
- **U**, **V**, and **W** — u , v , and w in units of meters.
- **Amp** — Data amplitudes in units which are proportional to Jansky (for data which are fully calibrated, the units should be in Jy).

- **Phase** — Data phases in units of degrees.
- **Real** and **Imag** — The real and imaginary parts of the visibility in units which are proportional to Jansky (for data which are fully calibrated, the units should be Jy).
- **Flag** — Data which are flagged have `Flag = 1`, whereas unflagged data are set to `Flag = 0`. Note that, to display flagged data, you will have to click on the **Plots > Display** tab and choose a **Flagged Points Symbol** (§ 3.4.1.6).
- **Azimuth** and **Ant-Azimuth** — Azimuth in units of degrees. **Azimuth** plots a fiducial value for the entire array, while **Ant-Azimuth** plots the azimuth for each individual antenna (their azimuths will differ by small amounts, because each antenna is located at a slightly different longitude, latitude, and elevation).
- **Elevation** and **Ant-Elevation** — Elevation in units of degrees. **Elevation** is a representative value for the entire array, while **Ant-Elevation** is the elevation for each individual antenna (their elevations will differ by small amounts, because each antenna is located at a slightly different longitude, latitude, and elevation).
- **HourAngle** — Hour angle in units of hours. This is a fiducial value for the entire array.
- **ParAngle** and **Ant-ParAng** — Parallax angle in units of degrees. **ParAngle** is the fiducial parallax angle for all antennae in the array, while **Ant-ParAng** plots the parallax angle for each individual antenna (their parallax angles will differ by small amounts, because each antenna is located at a slightly different longitude, latitude, and elevation).
- **Row** — Data row number. A row number corresponds to a unique time, baseline, and spectral window in the measurement set.
- **FlagRow** — In some tasks, if a whole data row is flagged, then `FlagRow` will be set to 1 for that row. Unflagged rows have `FlagRow = 0`. However, note that some tasks (like `plotms`) may flag a row, but *not* set `FlagRow = 1`. It is probably better to plot `Flag` than `FlagRow` for most applications.

If the data axis selected from the drop-down menu is already stored in the cache (therefore implying that plotting will proceed relatively quickly), an “X” will appear in the checkbox next to **In Cache?**.

For relevant data axes like **Amp** and **Phase**, the user will be presented with the option to plot raw data or calibrated data. This can be selected via a drop-down menu called **Data Column**, located directly under the drop-down menu for X or Y Axis selection (see the Y axis in Figure 3.2). To plot raw data, select “data”; to plot calibrated data, select “corrected”. Note that this choice will only have an impact on a plot if a calibration table has been applied to the measurement set (see `applycal`; § [WHICH SECTION?]).

If a data model has been applied to the measurement set (e.g., with `setjy`, § [WHICH SECTION?]), it can be plotted by selecting “model” from the **Data Column** menu. Finally, to plot the differences between the calibrated data and the model, select “residual” from **Data Column**.

3.4.1.3 Tools

Various tools—selectable as icon buttons at the bottom of the `plotms` window—can be used to zoom, edit, and locate data. The icon buttons can be seen at the bottom of Figures 3.1 and 3.2, and are, from left to right:

- **Zoom** — The “magnifying glass” button (1st on left) lets you draw a box around a region of the plot (left-click on one corner of the box, and drag the mouse to the opposite corner of the desired box), and then zooms in on this box.
- **Pan** — The “four-arrow” button (2nd from left) lets you pan around a zoomed plot.
- **Annotate** — The 3rd button from the left is chosen from a drop-down menu to either **Annotate Text** (“T with a green diamond” button) or **Annotate Rectangle** (“pencil” button). In the **Annotate Text** environment, click on a location in the plot where text is desired; a window will pop up, allowing you to type text in it. When you click the **OK** button, this text will appear on the plot. **Annotate Rectangle** simply lets you draw a box on the plot by left-clicking and dragging the mouse. By clicking on the **Annotator** tab near the top of the `plotms` window, different fonts, colors, line styles, etc. can be selected for annotations.
- **Home** — The “house” button (5th from left) returns to the original zoom level.
- **Stack Back** and **Stack Forward** — The left and right arrow buttons (4th and 6th from left) step through the zoom settings you’ve visited.
- **Mark Regions** — The “box with a green diamond” button (7th from left) lets you mark a region for flagging, unflagging, or locating. Left-click on one corner of the desired region, and then drag the mouse to set the opposite corner of the region. You can mark multiple boxes before performing an operation on them.
- **Clear Regions** — Clicking on the “box with a red circle” button (8th from left) will clear *all* regions which have been marked using **Mark Regions**.
- **Flag** — Click on the “flag” button (9th from left) to flag all points in the marked regions.
- **Unflag** — Click on the “crossed-out flag” button (10th from left) to unflag any flagged points that would be in the marked regions (even if invisible).
- **Locate** — The “magnifying glass on a sheet of paper” button (11th from left) will print out information to the command line about points in the marked regions.
- **Hold Drawing** — If the “hold drawing” button (rightmost, or 12th from left) is depressed, and if new plot axes are selected from the **Plots > Axes** tab, these new data will be cached but not plotted. When the button is clicked on again and un-depressed, it will automatically plot the data that was last requested. This can be particularly useful when changing the size of the `plotms` window.

There are two relevant options under the **Options** tab near the top of the `plotms` window. The **when changing plot axes, clear any existing regions or annotations** checkbox determines when regions and annotation are deleted from the plot. The **Tool Button Style** drop-down menu determines if icons and/or text represent the buttons at the bottom of the `plotms` window.

It is possible to hide these icons by going to the **View > Toolbars** menu at the top of the `plotms` window and un-depressing the **Tools** option (except for **Hide Drawing**, which is hidden by clicking on **View > Toolbars > Display**). In addition, the above tools can also be accessed by clicking on the **Tools** tab near the top of the `plotms` window (just below the **View** menu).

The **Tools** tab also enables one additional tool, the **Tracker**. To use **Tracker**, click on the **Hover** and/or **Display** checkbox, and place your mouse over the plot. **Tracker** will output the X and Y position of your mouse, either as text superimposed on the plot near your mouse (if **Hover** is selected) or in the blank window in the **Tools** tab (if **Display** is selected).

3.4.1.4 Interactive Flagging in `plotms`

Interactive flagging, on the principle of “see it — flag it”, is possible on the X-Y display of the data plotted by `plotms`. The user can use the cursor to mark one or more regions, and then flag, unflag, or list the data that falls in these zones of the display.

Using the row of icons buttons at the bottom of the `plotms` window (§ 3.4.1.3), click on the **Mark Regions** button (which will appear to depress), then mark a region by left-clicking and dragging the mouse (each click and drag will mark an additional region). You can get rid of all your regions by clicking on the **Clear Regions**. Once regions are marked, you can then click on one of the other buttons to take action:

1. **Flag** — flag the points in the region(s),
2. **Unflag** — unflag flagged points in the region(s),
3. **Locate** — spew out a list of the points in the region(s) to the command line (Warning: this could be a long list!).

Figure 3.3 shows an example of marking regions and then clicking the **Flag** button. Whenever you click on a button, that action occurs without requiring an explicit disk-write. If you quit `plotms` and re-enter, you will see your previous edits.

A table with the name `<msname>.flagversions` (where `vis=<msname>`) will be created in the same directory if it does not exist already. It is recommended that you save important flagging stages using the `flagmanager` task (§ 3.2).

Flags can also be extended with options in the **Flagging** tab, found near the top of the `plotms` window. Flag extension enables the user to plot a subset of the data and extend the flagging to a wider set. In this release, the only functional extensions are over channel and correlation.

By checking the boxes next to **Extend Flags** and **Channel**, flagging will be extended to other channels in the same `spw` as the displayed point. For example, if `spw='0:0'` and channel 0 is displayed, then flagging will extend to all channels in `spw 0`.

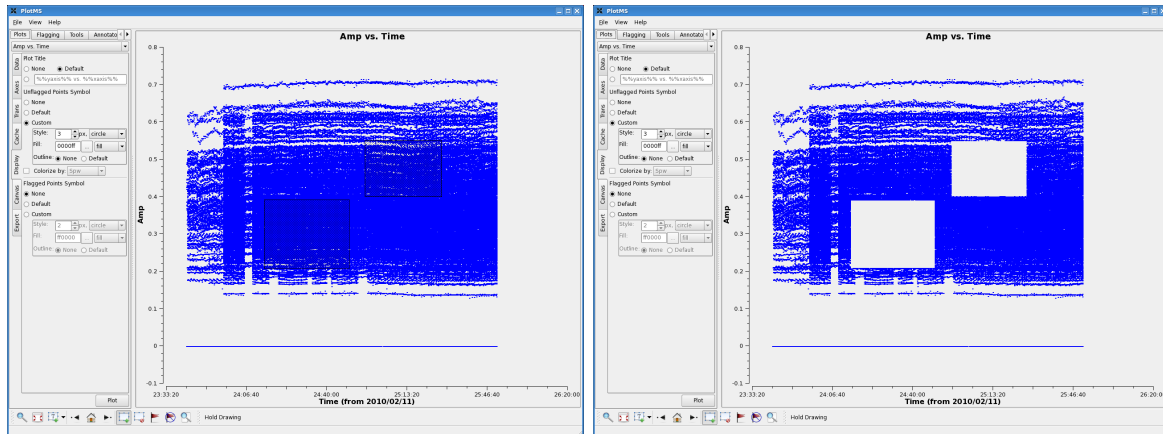


Figure 3.3: Plot of amplitude versus time, before (left) and after (right) flagging two marked regions. To unflag these regions, mark the two same regions and click the **Unflag** button.

By checking the boxes next to **Extend Flags** and **Correlation**, flags will be extended beyond the correlations displayed. Currently the only option is to extend to **All** correlations, implying that all correlations will be flagged, e.g. with RR displayed, the correlations RR, RL, LR, and LL will all be flagged.

WARNING: use of flag extensions may lead to deletion of much more data than desired. Be careful!

3.4.1.5 Averaging Data

The **Plots > Data** tab enables averaging of the data in order to increase signal-to-noise of the plotted points or to increase plotting speed. The options for **Averaging** are:

- **channel**
- **time**
- **all baselines** or **per antenna**
- **all spectral windows**
- **scalar**

The box next to a given **Averaging** mode needs to be checked for that averaging to take effect.

For example, to average n channels together, the user would click on the box next to **Channels** so that an “X” appears in it, and then type the number n in the empty box. When the user next clicks on **Plot**, every n channels will then be averaged together and the total number of channels plotted will be decreased by a factor of n .

Time averaging is a little trickier, as it is controlled by three fields. If the checkbox next to **Time** under **Averaging** is clicked on, a blank box with units of seconds will become active, along with two additional checkboxes: **Scan** and **Field**. If averaging is desired over a relatively short interval (say, 30 seconds, shorter than the scan length), a number can simply be entered into the blank box and, when the data are replotted, the data will be time averaged. Clicking on the **Scan** or **Field** checkbox in this case will have no impact on the time averaging.

These checkboxes become relevant if averaging over a relatively long time—say the entire observation, which consists of multiple scans—is desired. Regardless of how large a number is typed into the **Time** averaging blank box, only data within individual scans will be averaged together. In order to average data across scan boundaries, the **Scan** checkbox must be clicked on and the data replotted. Finally, clicking on the **Field** checkbox enables the averaging of multiple fields together in time

Clicking on the **All Baselines** checkbox will average all baselines in the array together. Alternatively, the **Per Antenna** box may be checked, which will average all baselines for a given antenna together. In this case, all baselines are represented twice; baseline 3-24 will contribute to the averages for both antenna 3 and antenna 24. This can produce some rather strange-looking plots if the user also selects on antenna—say, if the user requests to plot only antenna 0 and then averages **Per Antenna**. In this case, an average of all baselines including antenna 0 will be plotted, but each individual baseline including antenna 0 will also be plotted (because the presence of baselines 0-1, 0-2, 0-3, etc. trigger **Per Antenna** averaging to try and compute averages for antennae 1, 2, 3, etc. Therefore, baseline 0-1 will contribute to the average for antenna 0, but it will also singlehandedly *be* the average for antenna 1.)

Spectral windows can be averaged together by checking the box next to **All Spectral Windows**. This will result in, for a given channel n , all channels n from the individual spectral windows being averaged together.

Finally, the default mode is vector averaging, where the complex average is formed by averaging the real and imaginary parts of the relevant visibilities. If **Scalar** is chosen, then the amplitude of the average is formed by a scalar average of the individual visibility amplitudes.

When averaging, `plotms` will only include unflagged data in the average and will simply ignore flagged points that might otherwise be included in the average. This can occasionally have tricky consequences in plotting (see § 3.4.1.6).

3.4.1.6 Plot Symbols

Plot symbols are selected in the **Plots > Display** tab. Most fundamentally, the user can choose to plot unflagged data and/or flagged data. By default, unflagged data is plotted (the circle next to **Default** is checked under **Unflagged Points Symbol**), and flagged data is not plotted (the circle next to **None** is checked under **Flagged Points Symbol**). We note here that plotting flagged data on an averaged plot is undertaken at the user's own risk, as the distinction between flagged points and unflagged points becomes blurred if data are averaged over a dimension that is partially flagged. Take, for example, a plot of amplitude versus time where all channels are averaged together, but some channels have been flagged due to RFI spikes. In creating the average, `plotms`

will skip over the flagged channels and only use the unflagged ones. The averaged points will be considered unflagged, and the flagged data will not appear on the plot at all.

A selection of **None** produces no data points, **Default** results in data points which are small circles (blue for unflagged data and red for flagged data), and **Custom** allows the user to define a plot symbol. If **Custom** plot symbols are chosen, the user can determine the symbol size by typing a number in the blank box next to **px** or by clicking on the adjacent up or down arrows. Symbol shape can be chosen from the drop-down menu to be either “circle”, “square”, “diamond”, or “pixel” (note that “pixel” only has one possible size). Symbol color can be chosen by typing a hex color code in the blank box next to **Fill:** (e.g., “ff00ff”), or by clicking on the ... button and selecting a color from the pop-up GUI. The adjacent drop-down menu provides options for how heavily the plot symbol is shaded with this color, from heaviest to lightest: “fill”, “mesh1”, “mesh2”, “mesh3”, and “no fill”. Finally, the plot symbol can be outlined in black (if **Outline: Default** is checked) or not (if **Outline: None** is checked). Note that if “no fill” and **Outline: None** are selected, the plot symbols will be invisible.

Finally, unflagged data points can be given informative symbol colors using the **Colorize** parameter. By checking the box next to **Colorize** and selecting a data dimension from the drop-down menu, the data will be plotted with colors that vary along that dimension. For example, if “corr” is chosen from the **Colorize** menu, “RR”, “LL”, “RL”, and “LR” data will each be plotted with a different color. Note that, currently, **colorize** and plotting flagged data appear to be incompatible; a plot can only include one of these special features at a time [TRUE?].

3.4.1.7 Summarizing Data

Information about the measurement set can be obtained from within `plotms` by clicking on the **Summary** button, found at the bottom of the **Plots > Data** tab window. If “All” is chosen from the pull-down menu next to **Summary**, `listobs`-style output about scans, correlator configurations, and antennae will be written to the command line from which `plotms` was started. For more detail, click on the **Verbose** checkbox. For a specific subset of the data, choose a selection from the pull-down menu like “Antenna” or “Field”.

3.4.1.8 Defining Frequency and Velocity

If the user plans to plot **Frequency**, the reference frame must be defined. By default, the plotted frequency is simply that observed at the telescope. However, transformations can be made by choosing a **Frame** from the drop-down menu in the **Plots > Trans** tab. Frequency reference frames can be chosen to be:

- **LSRK** — local standard of rest (kinematic)
- **LSRD** — local standard of rest (dynamic)
- **BARY** — barycentric
- **GEO** — geocentric

- **TOPO** — topocentric
- **GALACTO** — galactocentric
- **LGROUP** — Local Group
- **CMB** — cosmic microwave background dipole

Velocity is affected by the user's choice of **Frame**, but it is also impacted by the choice of velocity definition and spectral line rest frequency. The velocity definition is chosen from the **Velocity Defn** drop-down menu in the **Plots > Trans** tab, offering selections of **Radio**, **True**, or **Optical**.

For more information on frequency frames and spectral coordinate systems, see the paper by Greisen et al. (A&A, 446, 747, 2006) ¹.

Finally, the spectral line's rest frequency in units of MHz should be typed into the blank box next to **Rest Freq** in the **Plots > Trans** tab. You can use the `me.spectralline` tool method to turn transition names into frequencies

```
CASA <16>: me.spectralline('HI')
Out[17]:
{'m0': {'unit': 'Hz', 'value': 1420405751.786},
 'refer': 'REST',
 'type': 'frequency'}
```

For a list of known lines in the CASA measures system, use the toolkit command `me.linelist()`. For example:

```
CASA <21>: me.linelist()
Out[21]: 'HI H186A H185A H184A H183A H182A H181A H180A H179A H178A H177A H176A H175A
H174A H173A H172A H171A H170A H169A H168A H167A H166A H165A H164A H163A H162A H161A H160A...
He182A He181A He180A He179A He178A He177A He176A He175A He174A He173A He172A He171A He170A
He169A He168A He167A He166A He165A He164A He163A He162A He161A He160A He159A He158A He157A...
C186A C185A C184A C183A C182A C181A C180A C179A C178A C177A C176A C175A C174A C173A C172A
C171A C170A C169A C168A C167A C166A C165A C164A C163A C162A C161A C160A C159A C158A C157A...
NH3_11 NH3_22 NH3_33 NH3_44 NH3_55 NH3_66 NH3_77 NH3_88 NH3_99 NH3_1010 NH3_1111 NH3_1212
OH1612 OH1665 OH1667 OH1720 OH4660 OH4750 OH4765 OH5523 OH6016 OH6030 OH6035 OH6049 OH13433
OH13434 OH13441 OH13442 OH23817 OH23826 CH3OH6.7 CH3OH44 H2O22 H2CO4.8 CO_1_0 CO_2_1 CO_3_2
CO_4_3 CO_5_4 CO_6_5 CO_7_6 CO_8_7 13CO_1_0 13CO_2_1 13CO_3_2 13CO_4_3 13CO_5_4 13CO_6_5
13CO_7_6 13CO_8_7 13CO_9_8 C180_1_0 C180_2_1 C180_3_2 C180_4_3 C180_5_4 C180_6_5 C180_7_6
C180_8_7 C180_9_8 CS_1_0 CS_2_1 CS_3_2 CS_4_3 CS_5_4 CS_6_5 CS_7_6 CS_8_7 CS_9_8 CS_10_9
CS_11_10 CS_12_11 CS_13_12 CS_14_13 CS_15_14 CS_16_15 CS_17_16 CS_18_17 CS_19_18 CS_12_19
SiO_1_0 SiO_2_1 SiO_3_2 SiO_4_3 SiO_5_4 SiO_6_5 SiO_7_6 SiO_8_7 SiO_9_8 SiO_10_9 SiO_11_10
SiO_12_11 SiO_13_12 SiO_14_13 SiO_15_14 SiO_16_15 SiO_17_16 SiO_18_17 SiO_19_18 SiO_20_19
SiO_21_20 SiO_22_21 SiO_23_22'
```

3.4.1.9 Shifting the Phase Center

The plot's phase center can be shifted in the **Plots > Trans** tab. Enter the X and Y shifts in units of arcseconds in the blank boxes under **Phase center shift**.

¹Also at <http://www.aoc.nrao.edu/~egreisen/scs.ps>

3.4.1.10 Plot Ranges

The X and Y ranges of the plot can be set in the **Plots > Axes** tab. By default, the circle next to **Automatic** will be checked, and the ranges will be auto-scaled. To define the range, click on the circle below **Automatic** and enter a minimum and maximum value in the blank boxes (as for the X Axis in Figure 3.2. Note that if identical values are placed in the blank boxes ($xmin=xmax$ and/or $ymin=ymax$), then the values will be ignored and a best guess will be made to auto-range that axis.

3.4.1.11 Plot Labels

The plot and axes labels which are displayed in the plot window are set in the **Plots > Canvas** tab. To change the plot title, under **Canvas Title**, click on the circle next to the blank box and enter the desired text (note that this is independent of the plot's title in the **Plot Selection** menu; see § 3.4.1.14). To change the X- and Y-axis labels, similarly click on the circles next to the blank boxes under **Show X Axis** and **Show Y Axis** and type the desired text in the blank box. To display these new labels, simply click the **Plot** button.

The user can determine the locations of axis labels in the **Plots > Axes** tab. The X-axis label switches from the bottom to the top of the plot depending on what is selected for **Attach to:**. Similarly for the Y-Axis, the user can choose to attach axis labels and tickmarks to the **Top** or **Bottom** (note that the axis labels have been attached to the **Bottom** and **Right** in Figure 3.2.

Finally, axis labels can be removed all together by unchecking the boxes next to **Show X Axis** and **Show Y Axis** on the **Plots > Canvas** tab.

3.4.1.12 Grid Lines

A grid of lines can be superimposed on the plot using **Grid Lines** in the **Plots > Canvas** tab. “Major” grid lines are drawn at the locations of major tick marks, while “minor” grid lines are drawn at minor tick marks.

Grid line colors, thicknesses, and styles are selected independently for the “major” and “minor” grid lines. Desired line thickness should be typed into the blank boxes just to the right of the **Major** and **Minor** labels. Colors are set by clicking on the ... buttons. The blank boxes to the left of the ... buttons will then contain the hex codes for the selected colors (e.g., “808080”). Line styles can also be selected from the drop-down menus to the right of ... buttons.

3.4.1.13 Legend

A plot symbol legend can be added to the plot by clicking on the checkbox next to **Legend** in the **Plots > Canvas** tab. However, given the current functionalities of `plotms`, a symbol legend is of very limited use. This option will become more relevant when overplotting capabilities are included in `plotms`.

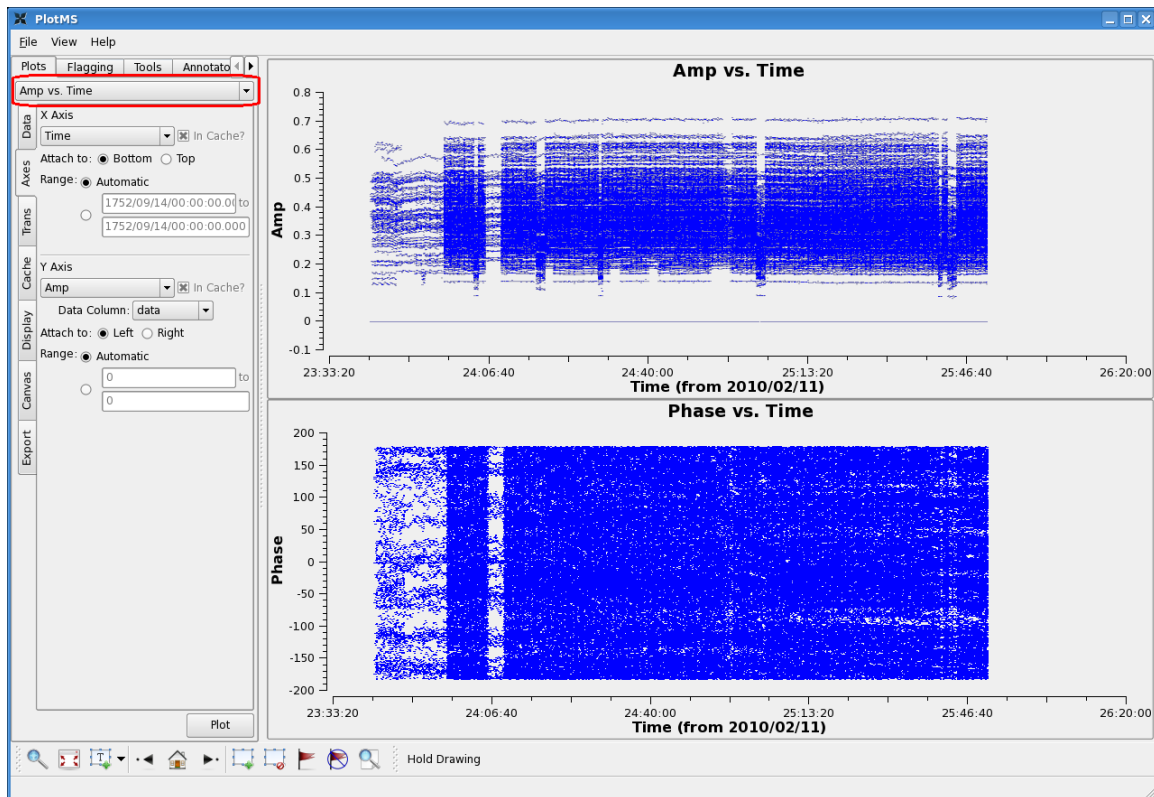


Figure 3.4: Multiple “single” plots displayed in a `plotms` window. The **Plot Selection** menu is circled in red (See § 3.4.1.14 for more detail).

3.4.1.14 New Plot Windows and Plot Selection

A new plot window can be opened using the drop-down menu which is located directly below the upper row of tabs (hereafter called the **Plot Selection** menu; it is labelled as “Amp vs. Time” and circled in red in Figure 3.4). To open a new window, select “New Single Plot” from this menu, and then click on the **Go** button which will appear after this selection. There will now be two plot windows, one on top of the other, as in Figure 3.4. The new window will be initially blank, and to plot in it, the measurement set must be re-loaded and re-selected upon (§ 3.4.1.1; then click the **Plot** button.

Both plot windows should now be listed in the **Plot Selection** drop-down menu. To return to editing the original plot, select it from this menu and click the **Edit** button which will appear on selection. The name of each plot in this drop-down menu can be changed using **Plot Title** in the **Plots > Axes** tab. Click on the circle next to the blank box under **Plot Title** and enter the desired text for the title. Note that this is, by default, the same as the plot title displayed in the plot window, but it is not required to be. The plot title in the plot window is determined by **Canvas Title** in the **Plots > Canvas** tab (see § 3.4.1.11).

The plot windows can be completely cleared by choosing “Clear Plotter” from the **Plot Selection** menu. Clicking the **Go** button will not only clear the plots themselves, but will also clear the plot parameters, loaded data, and the cache.

The **Plot Selection** menu also presents the user with a “New Multi Plot” option, but note that multi-plots are not functional in this release of CASA. Selecting “New Multi Plot” and clicking the **Go** button will currently crash `plotms`.

3.4.1.15 The Cache

Data dimensions which are loaded into the cache can be plotted relatively quickly. To see which data dimensions are loaded in the `plotms` cache, navigate to the **Plots > Cache** tab. The **Meta-Information Axes** are always loaded in the cache, and include data like **Channels** and **Time**.

The **Loaded Axes** are cached data dimensions, and have probably been recently plotted. To remove one of these axes from the cache, click on its name so that it is highlighted and then click the **Release** button.

Finally, the **Available Axes** are those data dimensions which are currently not stored in the cache. To load one of them to the cache, click on its name to highlight it and then click the **Load** button.

3.4.1.16 Iteration

`Plotms` does not include functionality for plot iteration in this release of CASA. We refer the reader to `plotxy` (§ 3.4.2.3) if plot iteration is desired,

3.4.1.17 Saving your plot

You can save a copy of a plot to file in the **Plots > Export** tab. Click the **Browse** button for a GUI-based selection of the directory and file name to which the plot will be saved. The file format can also be determined in this GUI by the suffix given to the filename: `.png` (PNG), `.jpg` (JPG), `.ps` (PS), or `.pdf` (PDF). Alternatively, the file format can be selected from the **Format** drop-down menu located just below the **Browse** button. In this case, `plotms` will add a suffix to the file name depending on the format chosen.

ALERT: The plot files produced by the PS and PDF options can be large and time-consuming to export. The JPG is the smallest.

The exported plot resolution can be manipulated using the **High Resolution**, **DPI**, and **Size** options.

Click on **Export** to create the file. Note that, if there is more than one plot displayed in the `plotms` window (§ 3.4.1.14), it will only export the currently selected plot.

3.4.1.18 Exiting plotms

To exit the `plotms` GUI, select **Quit** from the **File** menu at the top of the `plotms` window. You can also dismiss the window by killing it with the “X” on the frame.

Alternatively, you can just leave it alone, and `plotms` will keep running in the background. [WHAT HAPPENS IF THE MS FILE IS OVERWRITTEN OR EDITED ON DISK?]

3.4.2 Plotting and Editing using plotxy

ALERT: The `plotxy` code is fragile and slow, and is being replaced by the `plotms` (§ 3.4.1). We retain `plotxy` in this release as not all functionality is available yet in `plotms`.

`Plotxy` is a tool for visualizing and editing visibility data. Unlike `plotms`, it is useful in scripting, as it can non-interactively produce a hardcopy plot (see § 3.4.2.13). It also has `multiplot` (§ 3.4.2.8), iteration (§ 3.4.2.3), and overplotting (§ 3.4.2.4) functionality. Unlike `plotms` in the current release, `Access to plotms` is also provided through the `p1` tool. See below for a description of the `p1` tool functions.

To bring up this plotter use the `plotxy` task. The inputs are:

```
# plotxy :: X-Y plotter/interactive flagger for visibility data

vis          =      ''      # Name of input visibility
xaxis        =      'time'   # X-axis: def = 'time': see help for options
yaxis        =      'amp'    # Y-axis: def = 'amp': see help for options
  datacolumn =      'data'   # data (raw), corrected, model, residual (corrected - model)

selectdata   =      False   # Other data selection parameters
spw          =      ''      # spectral window:channels: ''==>all, spw='1:5~57'
field        =      ''      # field names or index of calibrators: ''==>all
averagemode  =      ''      # Select averaging type: 'vector', 'scalar'
restfreq     =      ''      # a frequency quanta or transition name. see help for options
extendflag   =      False   # Have flagging extend to other data points?
subplot      =      111     # Panel number on display screen (yxn)
plotsymbol   =      '.'     # Options include . : , o ^ v > < s + x D d 2 3 4 h H | _
plotcolor    =      'darkcyn' # Plot color
plotrange    =      [-1, -1, -1, -1] # The range of data to be plotted (see help)
multicolor   =      'corr'   # Plot in different colors: Options: none, both, chan, corr
selectplot   =      False   # Select additional plotting options (e.g, fontsize, title,etc)
overplot     =      False   # Overplot on current plot (if possible)
showflags    =      False   # Show flagged data?
interactive  =      True    # Show plot on gui?
figfile      =      ''      # ''= no plot hardcopy, otherwise supply name
async       =      False   # If true the taskname must be started using plotxy(...)
```

ALERT: The `plotxy` task expects all of the scratch columns to be present in the MS, even if it is not asked to plot the contents. If you get an error to the effect "Invalid Table operation:

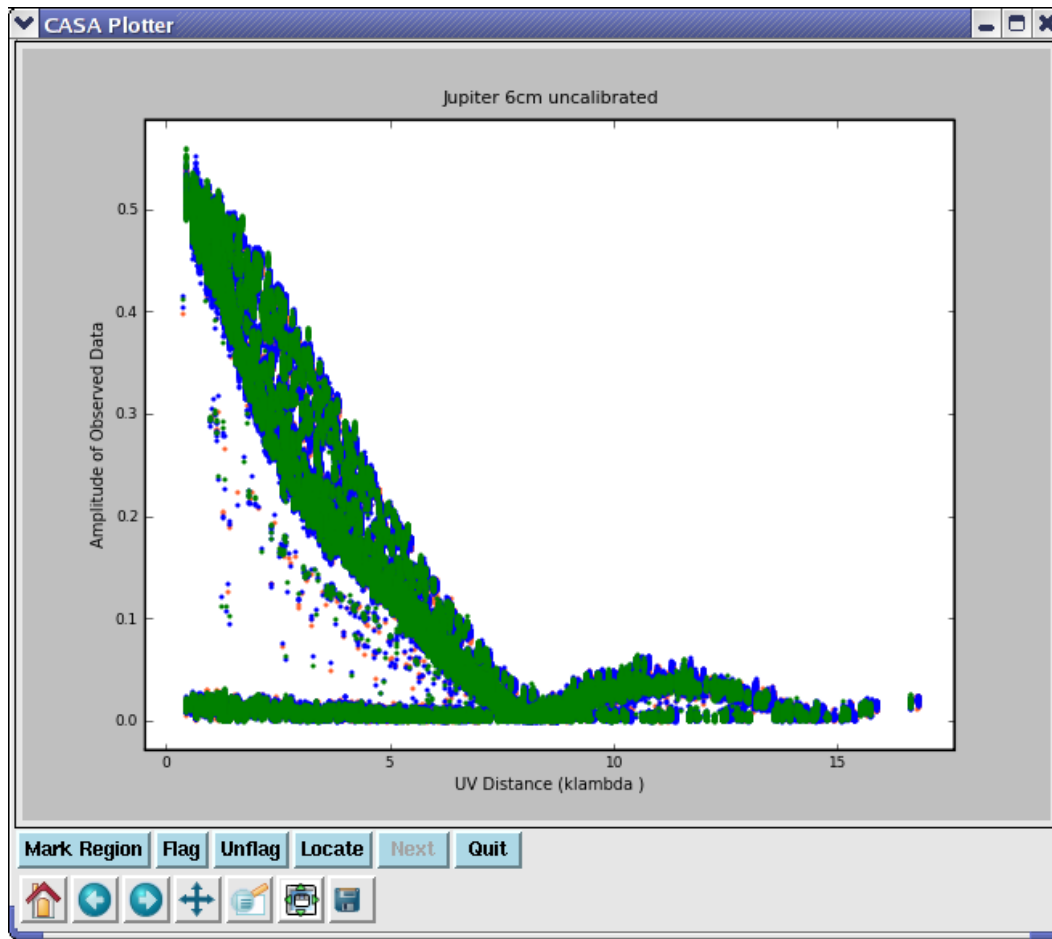


Figure 3.5: The plotxy plotter, showing the Jupiter data versus uv-distance. You can see bad data in this plot. The **bottom set of buttons** on the lower left are: 1,2,3) **Home, Back, and Forward**. Click to navigate between previously defined views (akin to web navigation). 4) **Pan**. Click and drag to pan to a new position. 5) **Zoom**. Click to define a rectangular region for zooming. 6) **Subplot Configuration**. Click to configure the parameters of the subplot and spaces for the figures. 7) **Save**. Click to launch a file save dialog box. The **upper set of buttons in the lower left** are: 1) **Mark Region**. Press this to begin marking regions (rather than zooming or panning). 2,3,4) **Flag, Unflag, Locate**. Click on these to flag, unflag, or list the data within the marked regions. 5) **Next**. Click to move to the next in a series of iterated plots. Finally, the **cursor readout** is on the bottom right.

Table: cannot add a column" then use `clearcal()` to force these columns to be made in the MS. Note that this will clear anything in all scratch columns (in case some were actually there and being used).

Setting `selectdata=True` opens up the selection sub-parameters:

```

selectdata      =      True   #   Other data selection parameters
  antenna       =      ''     #   antenna/baselines: ''==>all, antenna = '3,VA04'
  timerange     =      ''     #   time range: ''==>all
  correlation    =      ''     #   correlations: default = ''
  scan          =      ''     #   scan numbers: Not yet implemented
  feed          =      ''     #   multi-feed numbers: Not yet implemented
  array         =      ''     #   array numbers: Not yet implemented
  uvrange       =      ''     #   uv range''==>all; uvrange = '0~100kl' (default unit=meters)

```

These are described in § 2.3.

Averaging is controlled with the set of parameters

```

averagemode     =   'vector'  #   Select averaging type: vector, scalar
  timebin       =      '0'    #   Length of time-interval in seconds to average
  crossscans    =      False  #   Have time averaging cross scan boundaries?
  crossbls      =      False  #   have averaging cross over baselines?
  crossarrays   =      False  #   have averaging cross over arrays?
  stackspw      =      False  #   stack multiple spw on top of each other?
  width         =      '1'    #   Number of channels to average

```

See § 3.4.2.9 below for more on averaging.

You can extend the flagging beyond the data cell plotted:

```

extendflag      =   True     #   Have flagging extend to other data points?
  extendcorr    =      ''     #   flagging correlation extension type
  extendchan    =      ''     #   flagging channel extension type
  extendspw     =      ''     #   flagging spectral window extension type
  extendant     =      ''     #   flagging antenna extension type
  extendtime    =      ''     #   flagging time extension type

```

See § 3.4.2.11 below for more on flag extension.

The `restfreq` parameter can be set to a transition or frequency:

```

restfreq        =   'HI'     #   a frequency quanta or transition name. see help for options
  frame         =   'LSRK'   #   frequency frame for spectral axis. see help for options
  doppler       =   'RADIO'  #   doppler mode. see help for options

```

See § 3.4.2.12 below for more on setting rest frequencies and frames.

Setting `selectplot=True` will open up a set of plotting control sub-parameters. These are described in § 3.4.2.2 below.

The `interactive` and `figfile` parameters allow non-interactive production of hardcopy plots. See § 3.4.2.13 for more details on saving plots to disk.

The `iteration`, `overplot`, `plotrange`, `plotsymbol`, `showflags` and `subplot` parameters deserve extra explanation, and are described below.

For example:

```
plotxy(vis='jupiter6cm.ms',           # jupiter 6cm dataset
       xaxis='uvdist',                 # plot uv-distance on x-axis
       yaxis='amp',                     # plot amplitude on y-axis
       field='JUPITER',                 # plot only JUPITER
       selectdata=True,                 # open data selection
       correlation='RR,LL',             # plot RR and LL correlations
       selectplot=True,                 # open plot controls
       title = 'Jupiter 6cm uncalibrated') # give it a title
```

The plotter resulting from these settings is shown in figure 3.5.

ALERT: The `plotxy` task still has a number of issues. The averaging has been greatly speeded up in this release, but there are cases where the plots will be made incorrectly. In particular, there are problems plotting multiple `spw` at the same time. There are sometimes also cases where data that you have flagged in `plotxy` from averaged data is done so incorrectly. This task is under active development for the next cycle to fix these remaining problems, so users should be aware of this.

ALERT: Another known problem with (`plotxy`) is that it fails if the path to your working directory contains spaces in its name, e.g. `/users/smyers/MyTest/` is fine, but `/users/smyers/My Test/` is not!

3.4.2.1 GUI Plot Control

You can use the various buttons on the `plotxy` GUI to control its operation – in particular, to determine flagging and unflagging behaviors.

There is a standard row of buttons at the bottom. These include (left to right):

- **Home** — The “house” button (1st on left) returns to the original zoom level.
- **Step** — The left and right arrow buttons (2nd and 3rd from left) step through the zoom settings you’ve visited.
- **Pan** — The “four-arrow button” (4th from left) lets you pan in zoomed plot.
- **Zoom** — The most useful is the “magnifying glass” (5th from the left) which lets you draw a box and zoom in on the plot.
- **Panels** — The “window-thingy” button (second from right) brings up a menu to adjust the panel placement in the plot.

- **Save** – The “disk” button (last on right) saves a `.png` copy of the plot to a generically named file on disk.

In a row above these, there are a set of other buttons (left to right):

- **Mark Region** — If depressed lets you draw rectangles to mark points in the panels. This is done by left-clicking and dragging the mouse. You can Mark multiple boxes before doing something. Clicking the button again will un-depress it and forget the regions. ESC will remove the last region marked.
- **Flag** — Click this to Flag the points in a marked region.
- **Unflag** — Click this to Unflag any flagged point that would be in that region (even if invisible).
- **Locate** — Print out some information to the logger on points in the marked regions.
- **Next** — Step to the next plot in an iteration.
- **Quit** — Exit `plotcal`, clear the window and detach from the MS.

These buttons are shared with the `plotcal` tool.

3.4.2.2 The `selectplot` Parameters

These parameters work in concert with the native `matplotlib` functionality to enable flexible representations of data displays.

Setting `selectplot=True` will open up a set of plotting control sub-parameters:

```
selectplot      =      True   # Select additional plotting options (e.g, fontsize, title,etc)
  markersize    =      5.0   # Size of plotted marks
  linewidth     =      1.0   # Width of plotted lines
  skipnrows     =        1   # Plot every nth point
  newplot       =      False  # Replace the last plot or not when overplotting
  clearpanel    =      'Auto' # Specify if old plots are cleared or not
  title         =          '' # Plot title (above plot)
  xlabel        =          '' # Label for x-axis
  ylabel        =          '' # Label for y-axis
  fontsize      =     10.0   # Font size for labels
  windowsize    =      5.0   # Window size: not yet implemented
```

The `markersize` parameter will change the size of the plot symbols. Increasing it will help legibility when doing screen shots. Decreasing it can help in congested plots. The `linewidth` parameter will do similar things to the lines.

The `skipnrows` parameter, if set to an integer `n` greater than 1, will allow only every `n`th point to be plotted. It does this, as the name suggests, by skipping over whole rows of the MS, so beware (channels are all within the same row for a given `spw`). Be careful flagging on data where you have skipped points! Note that you can also reduce the number of points plotted via averaging (§ 3.4.2.9) or channel striding in the `spw` specification (§ 2.3.3).

The `newplot` toggle lets you choose whether or not the last layer plotted is replaced when `overplot=True`, or whether a new layer is added.

The `clearpanel` parameter turns on/off the clearing of plot panels that lie under the current panel layer being plotted. The options are: `'none'` (clear nothing), `'auto'` (automatically clear the plotting area), `'current'` (clear the current plot area only), and `'all'` (clear the whole plot panel).

The `title`, `xlabels`, and `ylabels` parameters can be used to change the plot title and axes labels.

The `fontsize` parameter is useful in order to enlarge the label fonts so as to be visible when making plots for screen capture, or just to improve legibility. Shrinking can help if you have lots of panels on the plot also.

The `windowsize` parameter is supposed to allow adjustments on the window size. **ALERT:** This currently does nothing, unless you set it below 1.0, in which case it will produce an error.

3.4.2.3 The iteration parameter

There are currently four iteration options available: `'field'`, `'antenna'`, and `'baseline'`. If one of these options is chosen, the data will be split into separate plot displays for each value of the iteration axis (e.g., for the VLA, the `'antenna'` option will get you 27 displays, one for each antenna).

An example use of iteration:

```
# choose channel averaging, every 5 channels
plotxy('n5921.ms', 'channel', subplot=221, iteration='antenna', width='5')
```

The results of this are shown in Figure 3.6. Note that this example combines the use of `width`, `iteration` and `subplot`.

NOTE: If you use `iteration='antenna'` or `'baseline'`, be aware if you have set `antenna` selection. You can also control whether you see auto-correlations or not using the appropriate syntax, e.g. `antenna='*&&*'` or `antenna='*&&&*'` (§ 2.3.4.1).

Inside the Toolkit:

For even more functionality, you can access the `pl` tool directly using Py-lab functions that allow one to annotate, alter, or add to any plot displayed in the `matplotlib` plotter (e.g. `plotxy`).

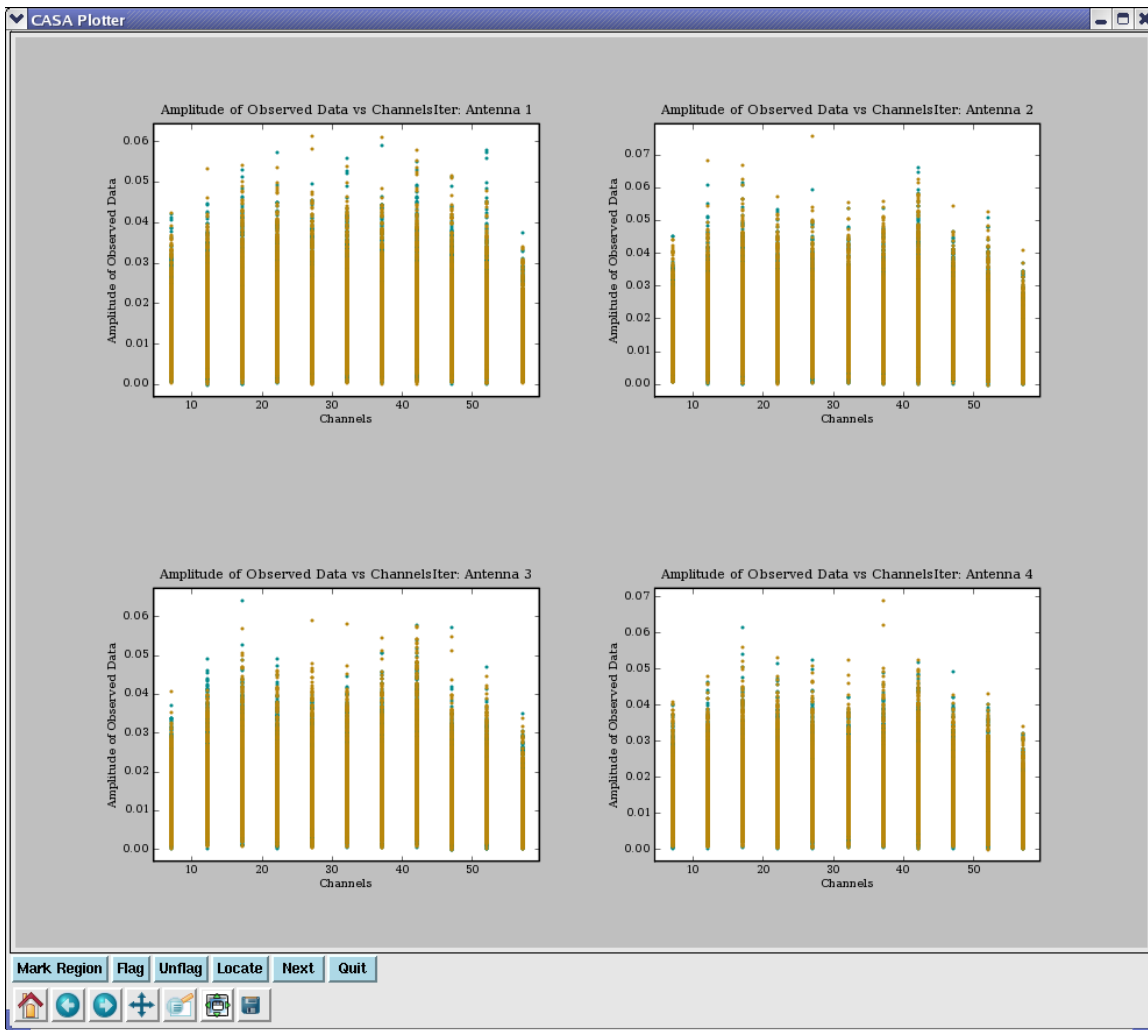


Figure 3.6: The plotxy iteration plot. The first set of plots from the example in § 3.4.2.3 with `iteration='antenna'`. Each time you press the **Next** button, you get the next series of plots.

3.4.2.4 The overplot parameter

The `overplot` parameter toggles whether the current plot will be overlaid on the previous plot or subpanel (via the `subplot` setting, § section:edit.plot.plotxy.subplot) or will overwrite it. The default is `False` and the new plot will replace the old.

The `overplot` parameter interacts with the `newplot` sub-parameter (see § 3.4.2.2).

See § 3.4.2.7 for an example using `overplot`.

3.4.2.5 The `plotrange` parameter

The `plotrange` parameter can be used to specify the size of the plot. The format is `[xmin, xmax, ymin, ymax]`. The units are those on the plot. For example,

```
plotrange = [-20,100,15,30]
```

Note that if `xmin=xmax` and/or `ymin=ymax`, then the values will be ignored and a best guess will be made to auto-range that axis. **ALERT:** Unfortunately, the units for the time axis must be in Julian Days, which are the plotted values.

3.4.2.6 The `plotsymbol` parameter

The `plotsymbol` parameter defines both the line or symbol for the data being drawn as well as the color; from the matplotlib online documentation (e.g., type `pl.plot?` for help):

The following line styles are supported:

```
-      : solid line
--     : dashed line
- .    : dash-dot line
:      : dotted line
.      : points
,      : pixels
o      : circle symbols
^      : triangle up symbols
v      : triangle down symbols
<      : triangle left symbols
>      : triangle right symbols
s      : square symbols
+      : plus symbols
x      : cross symbols
D      : diamond symbols
d      : thin diamond symbols
1      : tripod down symbols
2      : tripod up symbols
3      : tripod left symbols
4      : tripod right symbols
h      : hexagon symbols
H      : rotated hexagon symbols
p      : pentagon symbols
|      : vertical line symbols
_      : horizontal line symbols
steps : use gnuplot style 'steps' # kwarg only
```

The following color abbreviations are supported

```
b : blue
g : green
r : red
c : cyan
```

```

m : magenta
y : yellow
k : black
w : white

```

In addition, you can specify colors in many weird and wonderful ways, including full names 'green', hex strings '#008000', RGB or RGBA tuples (0,1,0,1) or grayscale intensities as a string '0.8'.

Line styles and colors are combined in a single format string, as in 'bo' for blue circles.

3.4.2.7 The showflags parameter

The `showflags` parameter determines whether *only* unflagged data (`showflags=False`) or flagged (`showflags=True`) data is plotted by this execution. The default is `False` and will show only unflagged "good" data.

Note that if you want to plot both unflagged and flagged data, in different colors, then you need to run `plotxy` twice using `overplot` (see § 3.4.2.4) the second time, e.g.

```

> plotxy(vis="myfile", xaxis='uvdist', yaxis='amp' )
> plotxy(vis="myfile", xaxis='uvdist', yaxis='amp', overplot=True, showflags=True )

```

3.4.2.8 The subplot parameter

The `subplot` parameter takes three numbers. The first is the number of y panels (stacking vertically), the second is the number of xpanels (stacking horizontally) and the third is the number of the panel you want to draw into. For example, `subplot=212` would draw into the lower of two panels stacked vertically in the figure.

An example use of subplot capability is shown in Fig 3.7. These were drawn with the commands (for the top, bottom left, and bottom right panels respectively):

```

plotxy('n5921.ms', 'channel',      # plot channels for the n5921.ms data set
      field='0',                  # plot only first field
      datacolumn='corrected',     # plot corrected data
      plotcolor='',              # over-ride default plot color
      plotsymbol='go',           # use green circles
      subplot=211)               # plot to the top of two panels

plotxy('n5921.ms', 'x',          # plot antennas for n5921.ms data set
      field='0',                  # plot only first field
      datacolumn='corrected',     # plot corrected data
      subplot=223,                # plot to 3rd panel (lower left) in 2x2 grid
      plotcolor='',              # over-ride default plot color
      plotsymbol='r.')           # red dots

plotxy('n5921.ms', 'u', 'v',     # plot uv-coverage for n5921.ms data set

```

```

field='0',           # plot only first field
datacolumn='corrected', # plot corrected data
subplot=224,        # plot to the lower right in a 2x2 grid
plotcolor='',       # over-ride default plot color
plotsymbol='b,')    # blue, somewhat larger dots
                    # NOTE: You can change the gridding
                    # and panel size by manipulating
                    # the ny x nx grid.

```

See also § 3.4.2.3 above, and Figure 3.6 for an example of channel averaging using iteration and subplot.

3.4.2.9 Averaging in plotxy

The averaging parameters and sub-parameters are:

```

averagemode      = 'vector' # Select averaging type: vector, scalar
timebin          = '0' # length of time in seconds to average, default='0', or: 'all'
crossscans       = False # have time averaging cross over scans?
crossbls         = False # have averaging cross over baselines?
crossarrays      = False # have averaging cross over arrays?
stackspw         = False # stack multiple spw on top of each other?
width            = '1' # number of channels to average, default: '1', or: 'all', 'allspw'

```

The choice of `averagemode` controls how the amplitudes are calculated in the average. The default mode is `'vector'`, where the complex average is formed by averaging the real and imaginary parts of the relevant visibilities. If `'scalar'` is chosen, then the amplitude of the average is formed by a scalar average of the individual visibility amplitudes.

Time averaging is effected by setting the `timebin` parameter to a value larger than the integration time. Currently, `timebin` takes a string containing the averaging time in seconds, e.g.

```
timebin = '60.0'
```

to plot one-minute averages.

Channel averaging is invoked by setting `width` to a value greater than 1. Currently, the averaging `width` is given as a number of channels.

By default, the averaging will not cross `scan` boundaries (as set in the import process). However, if `crossscans=True`, then averaging will cross scans.

Note that data taken in different sub-arrays are never averaged together. Likewise, there is no way to plot data averaged over `field`.

3.4.2.10 Interactive Flagging in plotxy

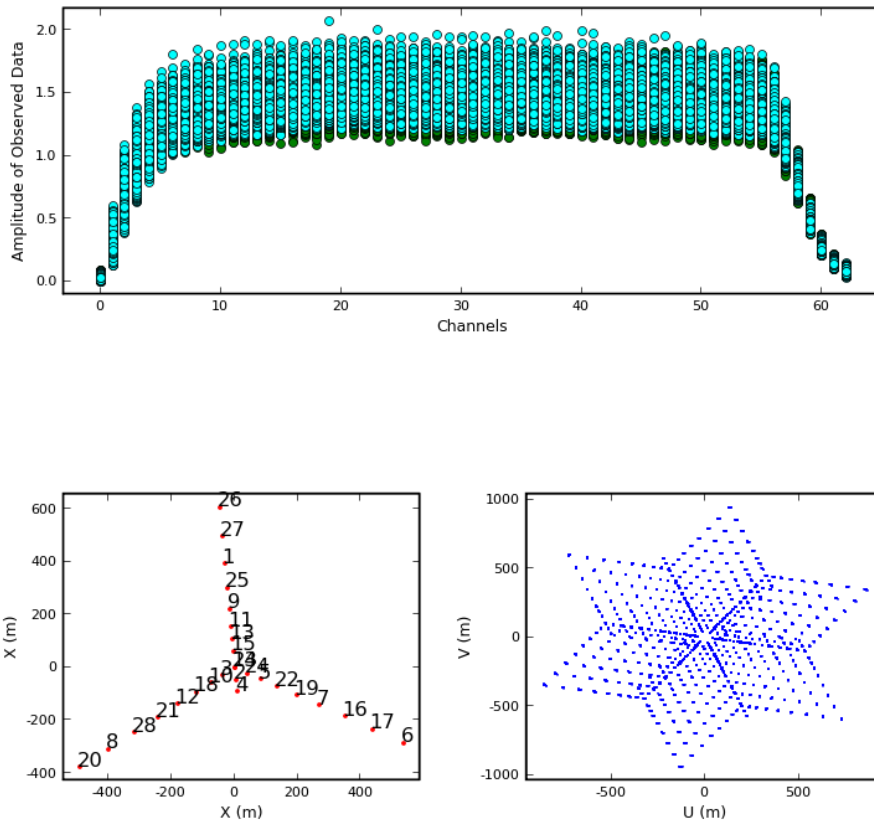


Figure 3.7: Multi-panel display of visibility versus channel (**top**), antenna array configuration (**bottom left**) and the resulting uv coverage (**bottom right**). The commands to make these three panels respectively are: 1) `plotxy('ngc5921.ms', xaxis='channel', datacolumn='data', field='0', subplot=211, plotcolor='', plotsymbol='go')` 2) `plotxy('ngc5921.ms', xaxis='x', field='0', subplot=223, plotsymbol='r.')`, 3) `plotxy('ngc5921.ms', xaxis='u', yaxis='v', field='0', subplot=224, plotsymbol='b.', figfile='ngc5921_multiplot.png')`.

Interactive flagging, on the principle of “see it — flag it”, is possible on the X-Y display of the data plotted by `plotxy`. The user can use the cursor to mark one or more regions, and then flag, unflag, or list the data that falls in these zones of the display.

Hint!

In the plotting environments such as `plotxy`, the ESC key can be used to remove the last region box drawn.

There is a row of buttons below the plot in the window. You can punch the **Mark Region** button (which will appear to depress), then mark a region by

left-clicking and dragging the mouse (each click and drag will mark an additional region). You can get rid of all your regions by clicking again on the **Mark Region** button (which will appear to un-depress), or you can use the ESC key to remove the last box you drew. Once regions are marked, you can then click on one of the other buttons to take action:

1. **Flag** — flag the points in the region(s),
2. **Unflag** — unflag flagged points in the region(s),
3. **Locate** — spew out a list of the points in the region(s) to the logger (Warning: this could be a long list!).

Whenever you click on a button, that action occurs without forcing a disk-write (unlike previous versions). If you quit `plotxy` and re-enter, you will see your previous edits.

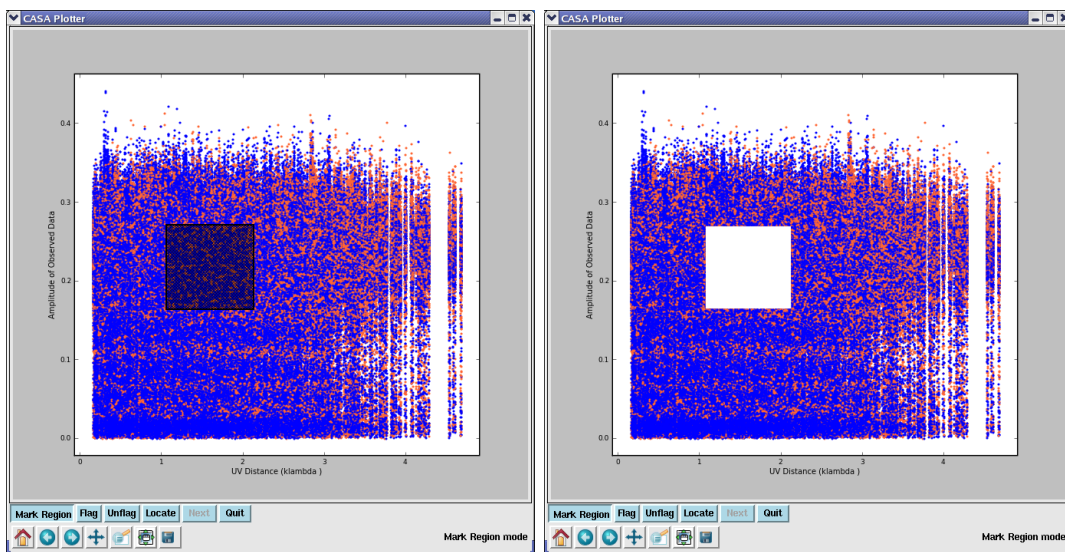


Figure 3.8: Plot of amplitude versus uv distance, before (left) and after (right) flagging two marked regions. The call was: `plotxy(vis='ngc5921.ms', xaxis='uvdist', field='1445*')`.

A table with the name `<msname>.flagversions` (where `vis=<msname>`) will be created in the same directory if it does not exist already.

It is recommended that you save important flagging stages using the `flagmanager` task (§ 3.2).

3.4.2.11 Flag extension in `plotxy`

Flag extension is controlled using `extendflag=T` and its sub-parameters:

```
extendflag      = True    # Have flagging extend to other data points?
```

```

extendcorr    =    ''    # flagging correlation extension type
extendchan    =    ''    # flagging channel extension type
extendspw     =    ''    # flagging spectral window extension type
extendant     =    ''    # flagging antenna extension type
extendtime    =    ''    # flagging time extension type

```

The use of `extendflag` enables the user to plot a subset of the data and extend the flagging to a wider set.

ALERT: Using the `extendflag` options will greatly slow down the flagging in `plotxy`. You will see a long delay after hitting the **Flag** button, with lots of logger messages as it goes through each flag. Fixing this requires a refactoring of `plotxy` which is underway starting in Patch 4 development.

Setting `extendchan='all'` will extend the flagging to other channels in the same `spw` as the displayed point. For example, if `spw='0:0'` and channel 0 is displayed, then flagging will extend to all channels in `spw 0`.

The `extendcorr` sub-parameter will extend the flagging beyond the correlations displayed. If `extendcorr='all'`, then all correlations will be flagged, e.g. with `RR` displayed `RR,RL,LR,LL` will be flagged. If `extendcorr='half'`, then the extension will be to those correlations in common with that show, e.g. with `RR` displayed then `RR,RL,LR` will be flagged.

Setting `extendspw='all'` will extend the flagging to all other `spw` for the selection. Using the same example as above, with `spw='0:0'` displayed, then channel 0 in ALL `spw` will be flagged. Note that use of `extendspw` could result in unintended behavior if the `spw` have different numbers of channels, or if it is used in conjunction with `extendchan`.

WARNING: use of the following options, particularly in conjunction with other flag extensions, may lead to deletion of much more data than desired. Be careful!

Setting `extendant='all'` will extend the flagging to all baselines that have antennas in common with those displayed and marked. For example, if `antenna='1&2'` is shown, then ALL baselines to BOTH antennas 1 and 2 will be flagged. Currently, there is no option to extend the flag to ONLY baselines to the first (or second) antenna in a displayed pair, so it is better to use `flagdata` to remove specific antennas.

Setting `extendtime='all'` will extend the flagging to all times matching the other selection or extension for the data in the marked region.

3.4.2.12 Setting rest frequencies in `plotxy`

The `restfreq` parameter can be set to a transition or frequency and expands to allow setting of frame information. For example,

```

restfreq      =    'HI'    # a frequency quanta or transition name. see help for options
frame        =    'LSRK'   # frequency frame for spectral axis. see help for options
doppler      =    'RADIO'  # doppler mode. see help for options

```

Examples of transitions include:

```
restfreq='1420405751.786Hz' # 21cm HI frequency
restfreq='HI' # 21cm HI transition name
restfreq='115.2712GHz' # CO 1-0 line frequency
```

For a list of known lines in the CASA `measures` system, use the toolkit command `me.linelist()`. For example:

```
CASA <14>: me.linelist()
Out[14]: 'C109A CI CII166A DI H107A H110A H138B H166A H240A H272A H2CO HE110A HE138B HI OH1612 OH1665 O
```

ALERT: The list of known lines in CASA is currently very restricted, and will be increased in upcoming releases (to include lines in ALMA bands for example).

You can use the `me.spectralline` tool method to turn transition names into frequencies

```
CASA <16>: me.spectralline('HI')
Out[17]:
{'m0': {'unit': 'Hz', 'value': 1420405751.786},
 'refer': 'REST',
 'type': 'frequency'}
```

(not necessary for this task, but possibly useful).

The `frame` sub-parameter sets the frequency frame. The allowed options can be listed using the `me.listcodes` method on the `me.frequency()` method, e.g.

```
CASA <17>: me.listcodes(me.frequency())
Out[17]:
{'extra': array([],
  dtype='|S1'),
 'normal': array(['REST', 'LSRK', 'LSRD', 'BARY', 'GEO', 'TOPO', 'GALACTO', 'LGROUP',
  'CMB'],
  dtype='|S8')}
```

The `doppler` sub-parameter likewise sets the Doppler system. The allowed codes can be listed using the `me.listcodes` method on the `me.doppler()` method,

```
CASA <18>: me.listcodes(me.doppler())
Out[18]:
{'extra': array([],
  dtype='|S1'),
 'normal': array(['RADIO', 'Z', 'RATIO', 'BETA', 'GAMMA', 'OPTICAL', 'TRUE',
  'RELATIVISTIC'],
  dtype='|S13')}
```

For most cases the 'RADIO' Doppler system is appropriate, but be aware of differences.

For more information on frequency frames and spectral coordinate systems, see the paper by Greisen et al. (A&A, 446, 747, 2006)².

²Also at <http://www.aoc.nrao.edu/~egreisen/scs.ps>

3.4.2.13 Printing from plotxy

There are two ways to get hardcopy plots in `plotxy`.

The first is to use the “disk save” icon from the interactive plot GUI to print the current plot. This will bring up a sub-menu GUI that will allow you to choose the filename and format. The allowed formats are `.png` (PNG), `.eps` (EPS), and `svg` (SVG). If you give the filename with a suffix (`.png`, `.eps`, or `svg`) it will make a plot of that type. Otherwise it will put a suffix on depending on the format chosen from the menu.

ALERT: The plot files produced by the EPS option can be large, and the SVG files can be very large. The PNG is the smallest.

The second is to specify a `figfile`. You probably want to disable the GUI using `interactive=False` in this case. The type of plot file that is made will depend upon the filename suffix. The allowed choices are `.png` (PNG), `.eps` (EPS), and `svg` (SVG).

This latter option is most useful from scripts. For example,

```
default('plotxy')
vis = 'ngc5921.ms'
field = '2'
spw = ''
xaxis = 'uvdist'
yaxis = 'amp'
interactive=False
figfile = 'ngc5921.uvplot.amp.png'
plotxy()
```

will plot amplitude versus uv-distance in PNG format. No `plotxy` GUI will appear.

ALERT: if you use this option to print to `figfile` with an `iteration` set, you will only get the first plot.

3.4.2.14 Exiting plotxy

You can use the **Quit** button to clear the plot from the window and detach from the MS. You can also dismiss the window by killing it with the X on the frame, which will also detach the MS.

You can also just leave it alone. The plotter pretty much keeps running in the background even when it looks like it’s done! You can keep doing stuff in the plotter window, which is where the `overplot` parameter comes in. Note that the `plotcal` task (§ 4.5.1) will use the same window, and can also overplot on the same panel.

If you leave `plotxy` running, beware of (for instance) deleting or writing over the MS without stopping. It may work from a memory version of the MS or crash.

3.4.2.15 Example session using plotxy

The following is an example of interactive plotting and flagging using `plotxy` on the Jupiter 6cm continuum VLA dataset. This is extracted from the script `jupiter6cm.usecase.py` available in the script area.

This assumes that the MS `jupiter6cm.usecase.ms` is on disk with `flagautocorr` already run.

```
default('plotxy')

vis = 'jupiter6cm.usecase.ms'

# The fields we are interested in: 1331+305,JUPITER,0137+331
selectdata = True

# First we do the primary calibrator
field = '1331+305'

# Plot only the RR and LL for now
correlation = 'RR LL'

# Plot amplitude vs. uvdist
xaxis = 'uvdist'
yaxis = 'amp'
multicolor = 'both'

# The easiest thing is to iterate over antennas
iteration = 'antenna'

plotxy()

# You'll see lots of low points as you step through RR LL RL LR
# A basic clip at 0.75 for RR LL and 0.055 for RL LR will work
# If you want to do this interactively, set
iteration = ''

plotxy()

# You can also use flagdata to do this non-interactively
# (see below)

# Now look at the cross-polar products
correlation = 'RL LR'

plotxy()

#-----
# Now do calibrator 0137+331
field = '0137+331'
correlation = 'RR LL'
```

```
xaxis = 'uvdist'
spw = ''
iteration = ''
antenna = ''

plotxy()

# You'll see a bunch of bad data along the bottom near zero amp
# Draw a box around some of it and use Locate
# Looks like much of it is Antenna 9 (ID=8) in spw=1

xaxis = 'time'
spw = '1'
correlation = ''

# Note that the strings like antenna='9' first try to match the
# NAME which we see in listobs was the number '9' for ID=8.
# So be careful here (why naming antennas as numbers is bad).
antenna = '9'

plotxy()

# YES! the last 4 scans are bad. Box 'em and flag.

# Go back and clean up
xaxis = 'uvdist'
spw = ''
antenna = ''
correlation = 'RR LL'

plotxy()

# Box up the bad low points (basically a clip below 0.52) and flag

# Note that RL,LR are too weak to clip on.

#-----
# Finally, do JUPITER
field = 'JUPITER'
correlation = ''
iteration = ''
xaxis = 'time'

plotxy()

# Here you will see that the final scan at 22:00:00 UT is bad
# Draw a box around it and flag it!

# Now look at whats left
correlation = 'RR LL'
xaxis = 'uvdist'
```

```

spw = '1'
antenna = ''
iteration = 'antenna'

plotxy()

# As you step through, you will see that Antenna 9 (ID=8) is often
# bad in this spw. If you box and do Locate (or remember from
# 0137+331) its probably a bad time.

# The easiset way to kill it:

antenna = '9'
iteration = ''
xaxis = 'time'
correlation = ''

plotxy()

# Draw a box around all points in the last bad scans and flag 'em!

# Now clean up the rest
xaxis = 'uvdist'
correlation = 'RR LL'
antenna = ''
spw = ''

# You will be drawing many tiny boxes, so remember you can
# use the ESC key to get rid of the most recent box if you
# make a mistake.

plotxy()

# Note that the end result is we've flagged lots of points
# in RR and LL. We will rely upon imager to ignore the
# RL LR for points with RR LL flagged!

```

3.4.3 Plotting antenna positions using plotants

This task is a simple plotting interface (to the `plotxy` functionality) to produce plots of the antenna positions (taken from the `ANTENNA` sub-table of the MS).

The inputs to `plotants` are:

```

# plotants :: Plot the antenna distribution in the local reference frame:
vis      =      '' # Name of input visibility file (MS)
figfile  =      '' # Save the plotted figure to this file
async    =      False #

```

3.5 Non-Interactive Flagging using flagdata

Task `flagdata` will flag the visibility data set based on the specified data selections, most of the information coming from a run of the `listobs` task (with/without `verbose=True`). Currently you can select based on any combination of:

- antennas (`antenna`)
- baselines (`antenna`)
- spectral windows and channels (`spw`)
- correlation types (`correlation`)
- field ids or names (`field`)
- uv-ranges (`uvrange`)
- times (`timerange`) or scan numbers (`scan`)
- antenna arrays (`array`)

and choose to flag, unflag, clip (`setclip` and sub-parameters), and remove the first part of each scan (`setquack`) and/or the autocorrelations (`autocorr`).

The inputs to `flagdata` are:

```
# flagdata :: Flag/Clip data based on selections:

vis           =          '' # Name of input visibility file
mode          = 'manualflag' # Mode (manualflag,shadow,quack,summary,autoflag,rfi)
autocorr      =          False # Flag autocorrelations
unflag        =          False # Unflag the data specified
clipexpr      = 'ABS RR' # Expression to clip on
clipminmax    =          [] # Range to use for clipping
clipcolumn    =          'DATA' # Data column to use for clipping
clipoutside   =          True # Clip outside the range, or within it
channelavg    =          False # Average over channels

spw           =          '' # spectral-window/frequency/channel
field         =          '' # Field names or field index numbers: ''==>all, field='0~2,3C286'
selectdata    =          True # More data selection parameters (antenna, timerange etc)
  antenna     =          '' # antenna/baselines: ''==>all, antenna = '3,VA04'
  timerange   =          '' # time range: ''==>all, timerange='09:14:0~09:54:0'
  correlation =          '' # Select data based on correlation
  scan        =          '' # scan numbers: ''==>all
  feed        =          '' # multi-feed numbers: Not yet implemented
  array       =          '' # (sub)array numbers: ''==>all
  uvrange     =          '' # uv range: ''==>all; uvrange = '0~100klambda', default units=meters

async        =          False # If true the taskname must be started using flagdata(...)
```

The default flagging mode is `'manualflag'`. See § 3.5.1.1 more more on this option.

The `mode='summary'` will print out a summary of the current state of flagging into the logger.

The `mode='quack'` will allow dropping of integrations from the beginning of scans. See § 3.5.1.2 for details.

The `mode='shadow'` option will allow shadowed data to be flagged, e.g. if it has not already during filling (§ 3.5.1.3).

There are also “autoflagging” modes `'autoflag'` and `'rfi'` available for testing (§ 3.5.1.4).

Note that often you want to apply many different flagging operations in a single pass through the data. For this, we have implemented a “list” mode for the `'manualflag'` selections where several `flagdata` task invocations can be combined into a single `flagdata` run by giving lists of parameters. This is possible for the modes `'manualflag'` and `'quack'`, only.

For example, the following three `flagdata` runs:

```
flagdata(vis='my.ms', mode='manualflag', selectdata=True, field='3', autocorr=True)
flagdata(vis='my.ms', mode='manualflag', selectdata=True, field='3', timerange = '6:0:0~6:23:00')
flagdata(vis='my.ms', mode='manualflag', selectdata=True, field='3', scan='0', spw='0:60;62;64')
```

can be combined into a single run by:

```
vis = 'my.ms'
mode = 'manualflag'
selectdata = True

field      = '3'
spw        = [ '' , '' , '' , '0:60;62;64' ]
autocorr   = [ True , False , False , False ]
timerange  = [ '' , '6:0:0~6:23:0', '' , '' ]
scan       = [ '' , '' , '' , '0' ]

flagdata()
```

Note that `field='3'` is equivalent to `field=['3','3','3']`.

3.5.1 Flag Antenna/Channels

The following commands give the results shown in Figure 3.9:

```
default{'plotxy'}
plotxy('ngc5921.ms', 'channel', iteration='antenna', subplot=311)
default('flagdata')
flagdata(vis='ngc5921.ms', antenna='0', spw='0:10~15')
default plotxy
plotxy('ngc5921.ms', 'channel', iteration='antenna', subplot=311)
```

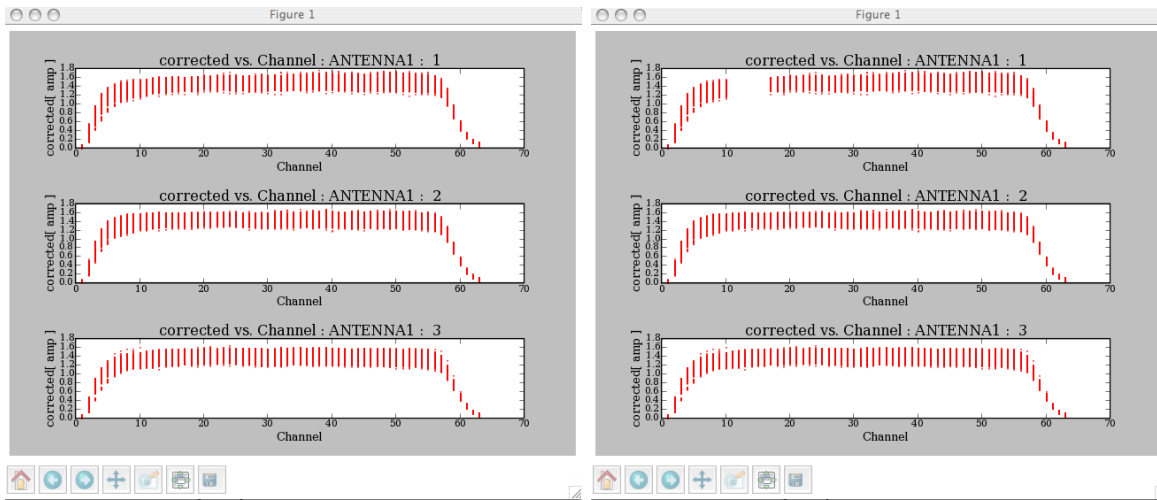


Figure 3.9: `flagdata`: Example showing before and after displays using a selection of one antenna and a range of channels. Note that each invocation of the `flagdata` task represents a cumulative selection, i.e., running `antenna='0'` will flag all data with antenna 0, while `antenna='0', spw='0:10 15'` will flag only those channels on antenna 0.

3.5.1.1 Manual flagging and clipping in `flagdata`

For `mode='manualflag'`, manual flagging and clipping is controlled by the sub-parameters:

```

mode           = 'manualflag' # Mode (manualflag,shadow,quack,summary,autoflag,rfi)
autocorr      = False       # Flag autocorrelations
unflag        = False       # Unflag the data specified
clipexpr      = 'ABS RR'    # Expression to clip on
clipminmax    = []         # Range to use for clipping
clipcolumn    = 'DATA'     # Data column to use for clipping
clipoutside   = True       # Clip outside the range, or within it
channelavg    = False       # Average over channels

```

The following commands give the results shown in Figure 3.10:

```

plotxy('ngc5921.ms', 'uvdist')
flagdata(vis='ngc5921.ms', clipexpr='LL', clipminmax=[0.0, 1.6], clipoutside=True)
plotxy('ngc5921.ms', 'uvdist')

```

The `channelavg` toggle (new in Version 3.0.0) is now available to (vector) average the data over all channels before doing the clipping test. This is most useful when flagging on phase stable or corrected data (e.g. after `applycal` and split to a new dataset).

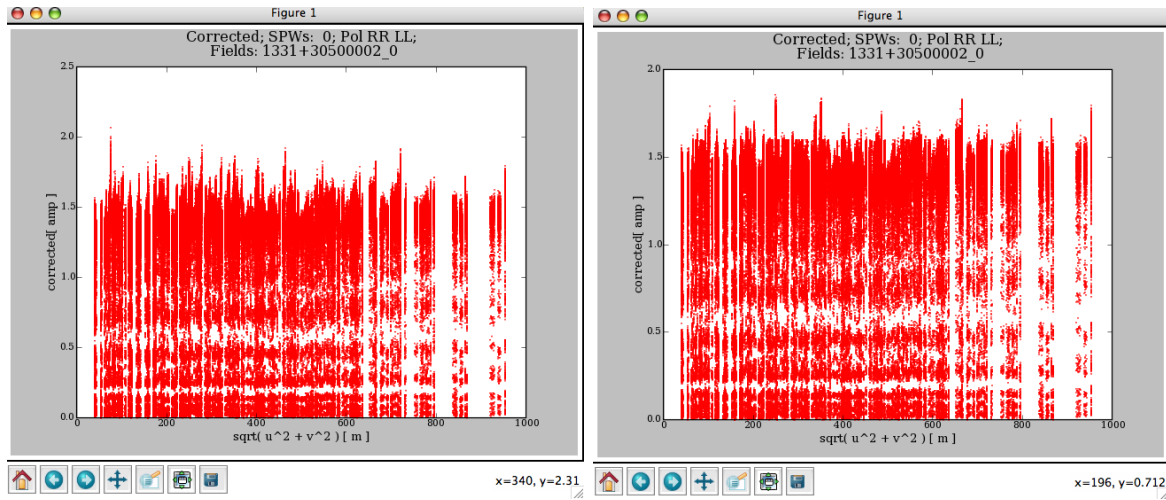


Figure 3.10: `flagdata`: Flagging example using the clip facility.

3.5.1.2 Flagging the beginning of scans

You can use the `mode='quack'` option to drop integrations from the beginning of scans (as in the AIPS task `QUACK`):

```

mode           = 'quack' # Mode (manualflag,shadow,quack,summary,autoflag,rfi)
autocorr       = False  # Flag autocorrelations
unflag         = False  # Unflag the data specified
quackinterval  = 0.0    # Quack n seconds from quackmode scan boundary
quackmode      = 'beg'  # 'beg' (start), 'endb' (end), 'end' (all but end), 'tail' (all but start)
quackincrement = False  # Flag incrementally in time?

```

Note that the time is measured from the first integration in the MS for a given scan, and this is often already flagged by the online system.

For example, assuming the integration time is 3.3 seconds (e.g. for VLA), then

```

mode = 'quack'
quackinterval = 14.0

```

will flag the first 4 integrations in every scan.

3.5.1.3 Flagging shadowed data with mode 'shadow'

Visibilities where one antenna is obscured by another antenna (typically at low elevations) can be flagged with the `'shadow'` mode:

```

mode          = 'shadow' # Mode (manualflag,shadow,quack,summary,autoflag,rfi)
diameter      = -1.0 # Effective diameter (m) to use. -1 ==>antenna diameter

```

Note that this can only flag data shadowed by antennas known in the MS (in the same subarray for example), not by antennas not in the dataset.

3.5.1.4 Autoflagging using modes 'autoflag' and 'rfi'

There are two “autoflagging” modes currently available in `flagdata`. **ALERT:** These are still experimental and under development for ALMA and EVLA commissioning.

The `mode='autoflag'` option uses an older autoflagging algorithm developed for `aips++`:

```

mode          = 'autoflag' # Mode (manualflag,shadow,quack,summary,autoflag,rfi)
algorithm     = 'timemed' # Autoflag algorithm (timemed,freqmed)
column       = 'DATA' # Data column to operate on.
expr         = 'ABS RR' # Expression to flag on
thr          = 5.0 # Flagging threshold (n sigma)
window       = 10 # Sliding median filter width

```

This algorithm passes median filters of a given width (`window`) in time (`timemed'`) or frequency (`freqmed'`) over the data and flagging outliers above a threshold (`'thr'`).

The `mode='rfi'` option uses an newer autoflagging algorithm:

```

mode          = 'rfi' # Mode (manualflag,shadow,quack,summary,autoflag,rfi)
clipcolumn    = 'DATA' # Data column to use for clipping
clipexpr      = 'ABS RR' # Expression to clip on
time_amp_cutoff = 4.0 # Flagging thresholds in units of sigma
freq_amp_cutoff = 3.0 # Flagging thresholds in units of sigma
freqlinefit   = False # Fit bandpass with a straight line (True), else piecewise polynomial
auto_cross    = 1 # Flag on cross and auto-correlations (0=autocor only)
num_time      = 400 # Number of time-steps in each chunk
start_chan    = 1 # Channel range start (1 based)
end_chan      = 2048 # Channel range end (1 based)
bs_cutoff     = 0.0 # Baseline tolerance
ant_cutoff    = 0.0 # Total autocorrelation amplitude threshold for a functional antenna
flag_level    = 1 # 0=no extend; 1=also one timestep before+after; 2= also one channel

```

3.6 Browse the Data

The `browsetable` task is available for viewing data directly (and handles all CASA tables, including Measurement Sets, calibration tables, and images). This task brings up the CASA Qt `casabrowser`, which is a separate program. You can launch this from outside `casapy`.

The default inputs are:


```
# browsetable :: Browse a table (MS, calibration table, image)

tablename      =      '' # Name of input table
async         =      False # If true the taskname must be started using browsetable(...)
```

Currently, its single input is the `tablename`, so an example would be:

```
browsetable('ngc5921.ms')
```

For an MS such as this, it will come up with a browser of the **MAIN** table (see Fig 3.11). If you want to look at sub-tables, use the **tab table keywords** along the left side to bring up a panel with the sub-tables listed (Fig 3.12), then choose (left-click) a table and **View:Details** to bring it up (Fig 3.13). You can left-click on a cell in a table to view the contents.

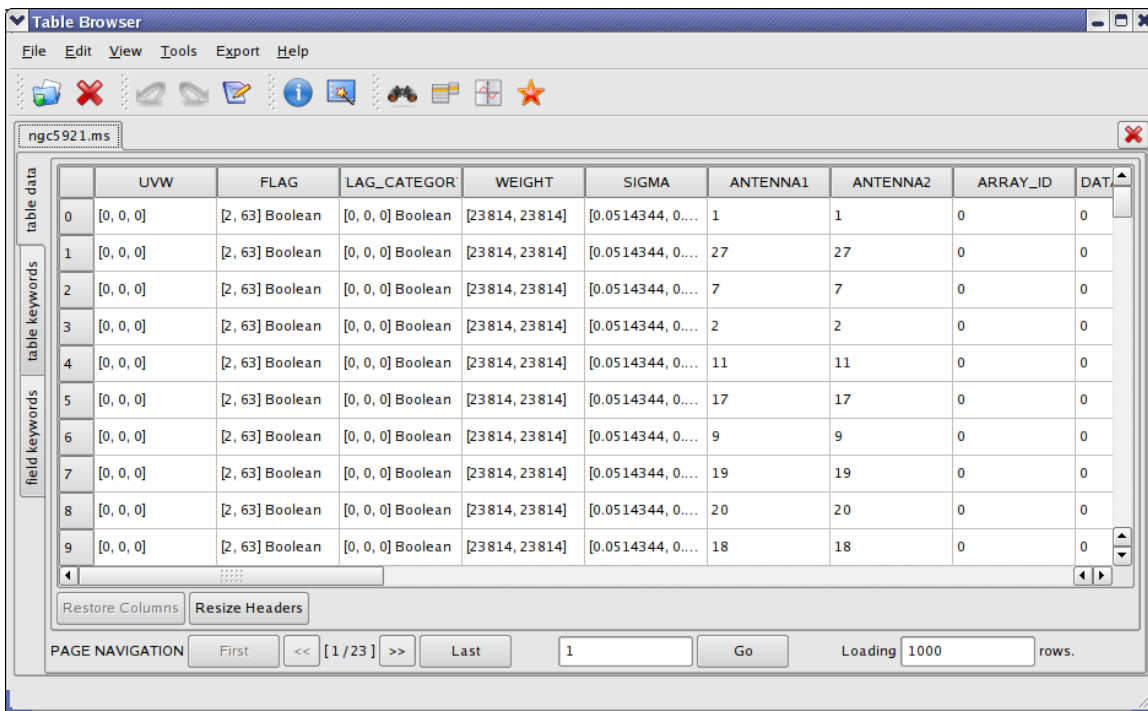


Figure 3.11: `browsetable`: The browser displays the main table within a frame. You can scroll through the data (x =columns of the **MAIN** table, and y =the rows) or select a specific page or row as desired. By default, 1000 rows of the table are loaded at a time, but you can step through the MS in batches.

Note that one useful feature is that you can Edit the table and its contents. Use the **Edit table** choice from the **Edit** menu, or click on the **Edit** button. Be careful with this, and make a backup copy of the table before editing!

The screenshot shows the 'Table Browser' window for a file named 'ngc5921.ms'. The window has a menu bar (File, Edit, View, Tools, Export, Help) and a toolbar with various icons. On the left, there are three vertical tabs: 'table data', 'table keywords', and 'field keywords'. The 'table keywords' tab is active, displaying a table with the following data:

	Keyword	Type	Value	Extra Information
1	MS_VERSION	Float	2	
2	ANTENNA	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/ANTENNA	Subtable has 28 rows.
3	DATA_DESCRIPTION	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/DATA_D...	Subtable has 1 rows.
4	FEED	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/FEED	Subtable has 28 rows.
5	FLAG_CMD	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/FLAG_C...	Subtable has no rows.
6	FIELD	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/FIELD	Subtable has 3 rows.
7	HISTORY	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/HISTORY	Subtable has 379 rows.
8	OBSERVATION	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/OBSER...	Subtable has 1 rows.
9	POINTING	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/POINTING	Subtable has no rows.
10	POLARIZATION	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/POLARI...	Subtable has 1 rows.
11	PROCESSOR	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/PROCE...	Subtable has no rows.
12	SPECTRAL_WINDOW	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/SPECTR...	Subtable has 1 rows.
13	STATE	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/STATE	Subtable has no rows.
14	SOURCE	Table	/home/sandrock/smyers/Testing/2008-03/ngc5921_regression/ngc5921.ms/SOURCE	Subtable has 3 rows.

Figure 3.12: `browsetable`: You can use the tab for Table Keywords to look at other tables within an MS. You can then double-click on a table to view its contents.

Use the `Close Tables` and `Exit` option from the **Files** menu to quit the `casabrowser`.

There are a lot of features in the `casabrowser` that are not fully documented here. Feel free to explore the capabilities such as plotting and sorting!

ALERT: You are likely to find that the `casabrowser` needs to get a table lock before proceeding. Use the `clearstat` command to clear the lock status in this case.

The screenshot shows a window titled "Table Browser" with a menu bar (File, Edit, View, Tools, Export, Help) and a toolbar. The main area displays a table for the file "ngc5921.ms" with the "SOURCE" table selected. The table has the following columns and data:

	DIRECTION	'ROPER_MOTIO	_LIBRATION_GRC	CODE	INTERVAL	NAME	NUM_LINES	SOURCE_ID	CTRLAL_W
0	[-2.74393, 0.5...	[0, 0]	-1		1.7976931348...	1331+305000...	1	0	0
1	[-2.42045, 0.1...	[0, 0]	-1		1.7976931348...	1445+099000...	1	1	0
2	[-2.2602, 0.08...	[0, 0]	-1		1.7976931348...	N5921_2	1	2	0

At the bottom of the window, there is a "PAGE NAVIGATION" section with buttons for "First", "<< [1/1] >>", and "Last", along with a "Go" button and a "Loading 1000 rows." indicator.

Figure 3.13: browsetable: Viewing the SOURCE table of the MS.

Chapter 4

Synthesis Calibration

This chapter explains how to calibrate interferometer data within the CASA task system. Calibration is the process of determining the complex correction factors that must be applied to each visibility in order to make them as close as possible to what an idealized interferometer would measure,

such that when the data is imaged an accurate picture of the sky is obtained. This is not an arbitrary process, and there is a philosophy behind the CASA calibration methodology (see § 4.2.1 for more on this). For the most part, calibration in CASA using the tasks is not too different than calibration in other packages such as AIPS or Miriad, so the user should not be alarmed by cosmetic differences such as task and parameter names!

Inside the Toolkit:

The workhorse for synthesis calibration is the `cb` tool.

4.1 Calibration Tasks

The standard set of `calibration` solving tasks (to produce calibration tables) are:

- `bandpass` — complex bandpass (B) calibration solving, including options for channel-binned or polynomial solutions (§ 4.4.2),
- `gaincal` — complex gain (G,T) calibration solving, including options for time-binned or spline solutions (§ 4.4.3),
- `polcal` — polarization calibration including leakage and angle (§ 4.4.5),
- `blcal` — *baseline-based* complex gain or bandpass calibration (§ 4.4.6).

There are helper tasks to create, manipulate, and explore calibration tables:

- `accum` — Accumulate incremental calibration solutions into a cumulative cal table (§ 4.5.5),
- `applycal` — Apply calculated calibration solutions (§ 4.6.1),

- `clearcal` — Re-initialize visibility data set calibration data (§ 4.6.3),
- `fluxscale` — Bootstrap the flux density scale from standard calibration sources (§ 4.4.4),
- `listcal` — List calibration solutions (§ 4.5.2),
- `plotcal` — Plot calibration solutions (§ 4.5.1),
- `setjy` — Compute the model visibility for a specified source flux density (§ 4.3.4),
- `smoothcal` — Smooth calibration solutions derived from one or more sources (§ 4.5.4),
- `split` — Write out new MS containing calibrated data from a subset of the original MS (§ 4.7.1).

There are some development versions of calibration and utility tasks that are recently added to the suite:

- `calstat` — Statistics of calibration solutions (§ 4.5.3),
- `cvel` — Regrid a spectral MS onto a new frequency channel system (§ 4.7.5),
- `gencal` — Create a calibration table from lists, including options for delays, antenna position errors, and amplitude and phase adjustment (§ 4.3.5),
- `hanningsmooth` — Apply a Hanning smoothing filter to spectral-line uv data (§ 4.7.2),
- `uvcontsub` — Carry out uv-plane continuum fitting and subtraction (§ 4.7.4),
- `uvmodelfit` — Fit a component source model to the uv data (§ 4.7.6),
- `uvsub` — Subtract the transform of a model image from the uv data (§ 4.7.3).

These are not yet full-featured, and may have only rudimentary controls and options.

The following sections outline the use of these tasks in standard calibration processes.

Information on other useful tasks and parameter setting can be found in:

- `listobs` — summary of a MS (§ 2.2.5),
- `listvis` — list data in a MS (§ 2.2.6),
- `plotms` — prototype next-generation X-Y plotting and editing (§ 3.4.1),
- `plotxy` — previous generation X-Y plotting and editing (§ 3.4.2),
- `flagdata` — non-interactive data flagging (§ 3.5),
- `data selection` — general data selection syntax (§ 2.3).

4.2 The Calibration Process — Outline and Philosophy

A work-flow diagram for CASA calibration of interferometry data is shown in Figure 4.1. This should help you chart your course through the complex set of calibration steps. In the following sections, we will detail the steps themselves and explain how to run the necessary tasks and tools.

This can be broken down into a number of discrete phases:

- **Prior Calibration** — set up previously known calibration quantities that need to be pre-applied, such as the flux density of calibrators, antenna gain-elevation curves, atmospheric models, delays, and antenna position offsets. Use the `setjy` task (§ 4.3.4) for flux densities and models, set the `gaincurve` (§ 4.3.2) and `opacity` (§ 4.3.3) parameters in subsequent tasks, and use `gencal` (§ 4.3.5) for delay and antenna position offsets;
- **Bandpass Calibration** — solve for the relative gain of the system over the frequency channels in the dataset (if needed), having pre-applied the prior calibration. Use the `bandpass` task (§ 4.4.2);
- **Gain Calibration** — solve for the gain variations of the system as a function of time, having pre-applied the bandpass (if needed) and prior calibration. Use the `gaincal` task (§ 4.4.3);
- **Polarization Calibration** — solve for any unknown polarization leakage terms (§ 4.4.5);
- **Establish Flux Density Scale** — if only some of the calibrators have known flux densities, then rescale gain solutions and derive flux densities of secondary calibrators. Use the `fluxscale` task (§ 4.4.4);
- **Manipulate, Accumulate, and Iterate** — if necessary, accumulate different calibration solutions (tables), smooth, and interpolate/extrapolate onto different sources, bands, and times. Use the `accum` (§ 4.5.5) and `smoothcal` (§ 4.5.4) tasks;
- **Examine Calibration** — at any point, you can (and should) use `plotcal` (§ 4.5.1) and/or `listcal` (§ 4.5.2) to look at the calibration tables that you have created;
- **Apply Calibration to the Data** — this can be forced explicitly by using the `applycal` task (§ 4.6.1), and can be undone using `clearcal` (§ 4.6.3);
- **Post-Calibration Activities** — this includes the determination and subtraction of continuum signal from line data, the splitting of data-sets into subsets (usually single-source), and other operations (such as model-fitting). Use the `uvcontsub` (§ 4.7.4), `split` (§ 4.7.1), and `uvmodelfit` (§ 4.7.6) tasks.

The flow chart and the above list are in a suggested order. However, the actual order in which you will carry out these operations is somewhat fluid, and will be determined by the specific data-reduction use cases you are following. For example, you may need to do an initial **Gain Calibration** on your bandpass calibrator before moving to the **Bandpass Calibration** stage. Or perhaps the polarization leakage calibration will be known from prior service observations, and can be applied as a constituent of Prior Calibration.

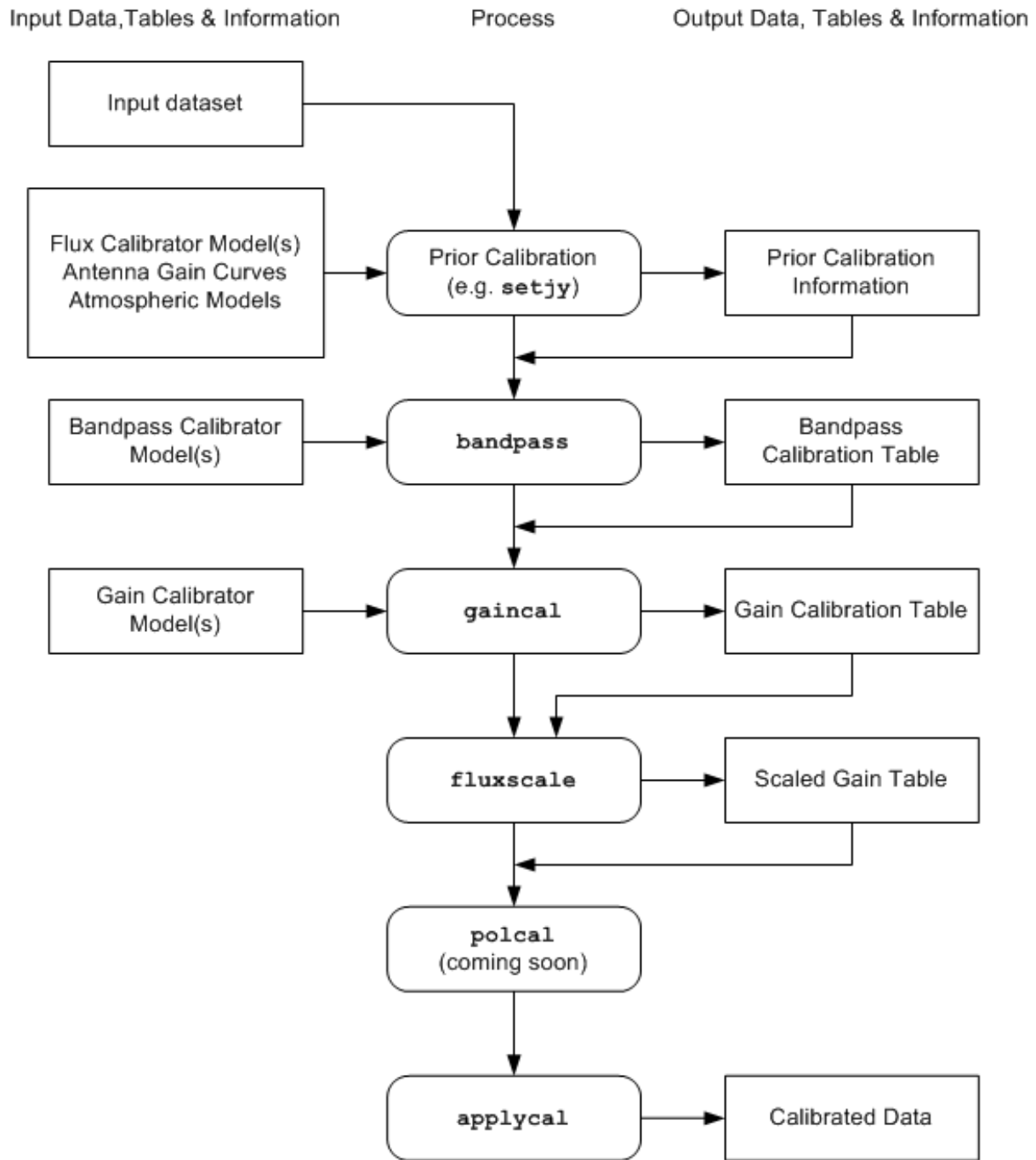


Figure 4.1: Flow chart of synthesis calibration operations. Not shown are use of table manipulation and plotting tasks `accum`, `plotcal`, and `smoothcal` (see Figure 4.2).

4.2.1 The Philosophy of Calibration in CASA

Calibration is not an arbitrary process, and there is a methodology that has been developed to carry out synthesis calibration and an algebra to describe the various corruptions that data might be subject to: the Hamaker-Bregman-Sault Measurement Equation (ME), described in Appendix E. The user need not worry about the details of this mathematics as the CASA software does that for you. Anyway, its just matrix algebra, and your familiar scalar methods of calibration (such as in AIPS) are encompassed in this more general approach.

There are a number of “physical” components to calibration in CASA:

- **data** — in the form of the Measurement Set (§ 2.1). The MS includes a number of columns that can hold calibrated data, model information, and weights;
- **calibration tables** — these are in the form of standard CASA tables, and hold the calibration solutions (or parameterizations thereof);
- **task parameters** — sometimes the calibration information is in the form of CASA task parameters that tell the calibration tasks to turn on or off various features, contain important values (such as flux densities), or list what should be done to the data.

At its most basic level, Calibration in CASA is the process of taking “uncalibrated” **data**, setting up the operation of calibration tasks using **parameters**, solving for new calibration **tables**, and then applying the calibration tables to form “calibrated” **data**. Iteration can occur as necessary, with the insertion of other non-calibration steps (e.g. “self-calibration” via imaging).

4.2.2 Keeping Track of Calibration Tables

The calibration tables are the currency that is exchanged between the calibration tasks. The “solver” tasks (`gaincal`, `bandpass`, `blcal`, `fringecal`) take in the MS (which may have a calibration model in the `MODEL_DATA` column from `setjy` or `ft`) and previous calibration tables, and will output an “incremental” calibration table (it increments the previous calibration, if any). This table can then be smoothed using `smoothcal` if desired.

You can accumulate the incremental calibration onto previous calibration tables with `accum`, which will then output a cumulative calibration table. This task will also interpolate onto a different time scale. See § 4.5.5 for more on accumulation and interpolation.

Figure 4.2 graphs the flow of these tables through the sequence

```
solve => smooth => accumulate
```

Note that this sequence applied to separate *types* of tables (e.g. 'B', 'G') although tables of other types can be previous calibration input to the solver.

The final set of cumulative calibration tables is what is applied to the data using `applycal`. You will have to keep track of which tables are the intermediate incremental tables, and which are

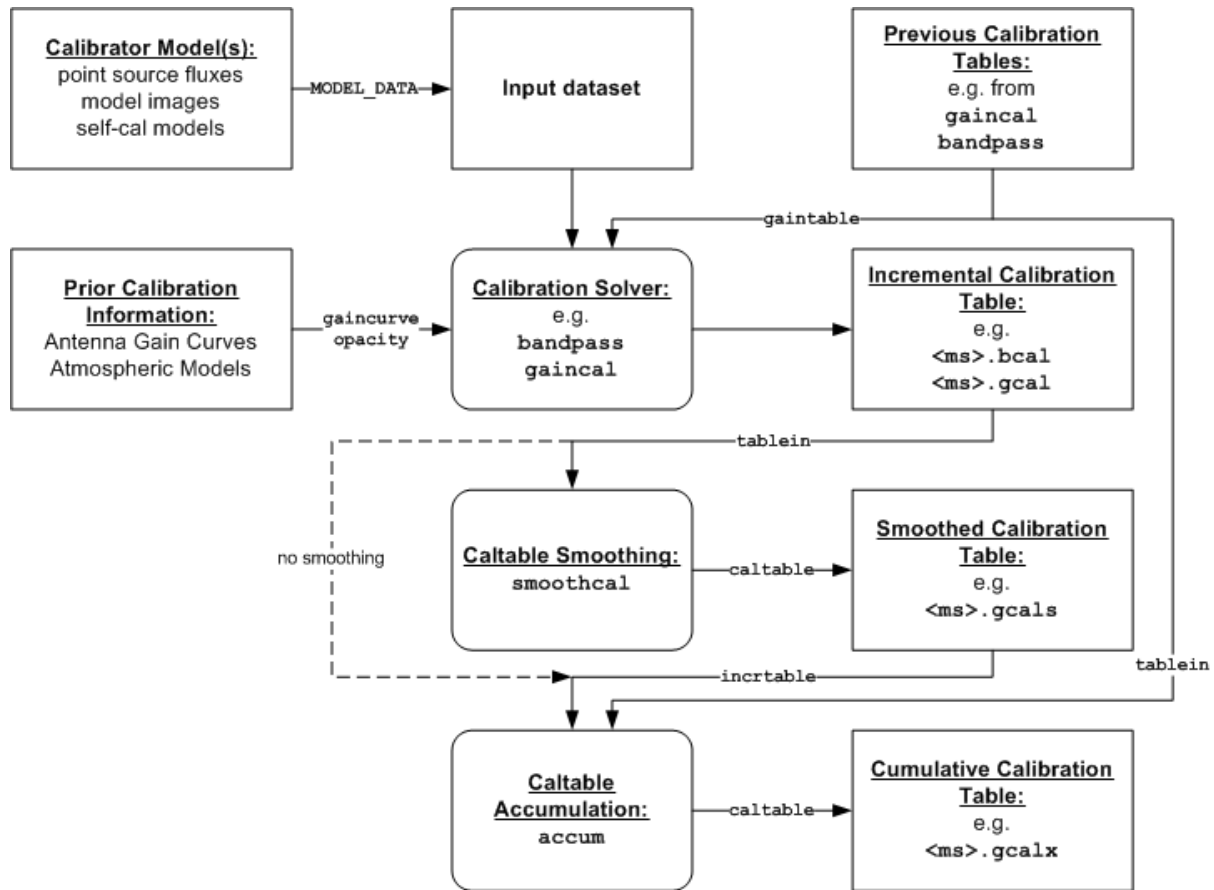


Figure 4.2: Chart of the table flow during calibration. The parameter names for input or output of the tasks are shown on the connectors. Note that from the output solver through the accumulator only a single calibration type (e.g. 'B', 'G') can be smoothed, interpolated or accumulated at a time. The final set of cumulative calibration tables of all types are then input to `applycal` as shown in Figure 4.1.

cumulative, and which were previous to certain steps so that they can also be previous to later steps until accumulation. This can be a confusing business, and it will help if you adopt a consistent table naming scheme (see Figure 4.2) for an example naming scheme).

4.2.3 The Calibration of VLA data in CASA

CASA supports the calibration of VLA data that is imported from the Archive through the `importvla` task. See § 2.2.2 for more information.

ALERT: Data taken both before and after the Modcomp turn-off in late June 2007 will be handled automatically by `importvla`. You do not need to set special parameters to do so, and it will obey

the scaling specified by `applytsys`.

You can also import VLA data in UVFITS format with the `importuvfits` task (§ 2.2.3.1). However, in this case, you must be careful during calibration in that some prior or previous calibrations (see below) may or may not have been done in AIPS and applied (or not) before export.

For example, the default settings of AIPS `FILLM` will apply VLA `gaincurve` and approximate (weather-based) atmospheric optical depth corrections when it generates the extension table `CL 1`. If the data is exported immediately using `FITTP`, then this table is included in the UVFITS file. However, CASA is not able to read or use the AIPS `SN` or `CL` tables, so that prior calibration information is lost and must be applied during calibration here (ie. using `gaincurve=True` and setting the `opacity` parameter).

On the other hand, if you apply calibration in AIPS by using the `SPLIT` or `SPLAT` tasks to apply the `CL` tables before exporting with `FITTP`, then this calibration will be in the data itself. In this case, you do not want to re-apply these calibrations when processing in CASA.

4.3 Preparing for Calibration

There are a number of “a priori” calibration quantities that may need to be applied to the data before further calibration is carried out. These include

- **system temperature correction** — turn correlation coefficient into correlated flux density (necessary for some telescopes),
- **gain curves** — antenna gain-elevation dependence,
- **atmospheric optical depth** — attenuation of the signal by the atmosphere, correcting for its elevation dependence.
- **flux density models** — establish the flux density scale using “standard” calibrator sources, with models for resolved calibrators,
- **delays** — antenna-based delay offsets,
- **antenna position errors** — offsets in the positions of antennas assumed during correlation.

These are pre-determined effects and should be applied (if known) before solving for other calibration terms. If unknown, then they will need to be solved for (or subsumed in other calibration such as bandpass or gains).

We now deal with these in turn.

4.3.1 System Temperature Correction

Some telescopes, including the EVLA and the VLBA, record the visibilities in the form of raw *correlation coefficient* with weights proportional to the number of bits correlated. The correlation coefficient is the fraction of the total signal that is correlated, and thus multiplication by the system temperature and the antenna gain (in Jy/K) will produce visibilities with units of correlated flux density. Note that the old VLA system did this initial calibration on-line, and ALMA will also provide some level of on-line calibration (TBD).

ALERT: There is as yet no mechanism available in `importvla` or in the calibration tasks to use the system temperature information provided by the VLA/EVLA on-line system to calibrate EVLA or VLBA data in raw form. This includes VLA data taken after the Modcomp turn-over in late June 2007. You may pass the data through AIPS first. You can also just forge ahead with standard calibration. The drawback to this is that short-term changes in T_{sys} which are not tracked by calibrator observations or self-calibration will remain in the data.

4.3.2 Antenna Gain-Elevation Curve Calibration

Large antennas (such as the 25-meter antennas used in the VLA and VLBA) have a forward gain and efficiency that changes with elevation. Gain curve calibration involves compensating for the effects of elevation on the amplitude of the received signals at each antenna. Antennas are not absolutely rigid, and so their effective collecting area and net surface accuracy vary with elevation as gravity deforms the surface. This calibration is especially important at higher frequencies where the deformations represent a greater fraction of the observing wavelength. By design, this effect is usually minimized (i.e., gain maximized) for elevations between 45 and 60 degrees, with the gain decreasing at higher and lower elevations. Gain curves are most often described as 2nd- or 3rd-order polynomials in zenith angle.

Gain curve calibration has been implemented in CASA for the VLA (only), with gain curve polynomial coefficients available directly from the CASA data repository. To make gain curve corrections for VLA data, set `gaincurve=True` for any of the calibration tasks.

ALERT: The `gaincurve` parameter must be supplied to any calibration task that allows pre-application of the prior calibration (e.g. `bandpass`, `gaincal`, `applycal`). This should be done consistently through the calibration process. In future updates we will add the capability to `gencal` (§ 4.3.5) to create a calibration table for this.

For example, to pre-apply the `gaincurve` during gain calibration:

```
gaincal('data.ms', 'cal.G0', gaincurve=True, solint=0., refant=11)
```

NOTE: Set `gaincurve=False` if you are not using VLA data.

The gain curve will be calculated per timestamp. Upon execution of a calibration task (e.g., `gaincal`, `bandpass`, `applycal`, etc.), the gain curve data appropriate to the observing frequencies will be automatically retrieved from the data repository and applied.

ALERT: Currently, gain-curves for VLA are built into the CASA system and this is what is applied when `gaincurve=True`. Therefore, the application of the gain-curves, if `gaincurve=True`, is allowed only if the VLA is set as the telescope of observation in the MS, otherwise an error will be generated. Set `gaincurve=False` if you are not using VLA data. A general mechanism for incorporating gaincurve information for other arrays will be made available in future releases. The gain-curve information available for the VLA is time-dependent (on timescales of months to years, at least for the higher frequencies), and CASA will automatically select the date-appropriate gain curve information. Note, however, that the time-dependence was poorly sampled prior to 2001, and so gain curve corrections prior to this time should be considered with caution. Similarly, gain curves have not yet been measured for antennas that have been physically modified to meet EVLA specifications. We will include gain curves for EVLA antennas when those are measured and become available.

4.3.3 Atmospheric Optical Depth Correction

The troposphere is not completely transparent. At high radio frequencies (>15 GHz), water vapor and molecular oxygen begin to have a substantial effect on radio observations. According to the physics of radiative transmission, the effect is threefold. First, radio waves from astronomical sources are absorbed (and therefore attenuated) before reaching the antenna. Second, since a good absorber is also a good emitter, significant noise-like power will be added to the overall system noise. Finally, the optical path length through the troposphere introduces a time-dependent phase error. In all cases, the effects become worse at lower elevations due to the increased air mass through which the antenna is looking. In CASA, the opacity correction described here compensates only for the first of these effects, tropospheric attenuation, using a plane-parallel approximation for the troposphere to estimate the elevation dependence.

Opacity corrections are a component of calibration type 'T'. To make opacity corrections in CASA, an estimate of the zenith opacity is required (see observatory-specific chapters for how to measure zenith opacity). This is then supplied to the `opacity` parameter in the calibration tasks.

ALERT: The `opacity` parameter must be supplied to any calibration task that allows pre-application of the prior calibration (e.g. `bandpass`, `gaincal`, `applycal`). This should be done consistently through the calibration process. In future updates we will add the capability to `gencal` (§ 4.3.5) to create a calibration table for this. Furthermore, you currently can only supply a single value of `opacity`, which will then be pre-applied to whatever calibration task that you set it in. Generalizations to antenna- and time-dependent opacities, including derivation (from weather information) and solving (directly from the visibility data) capabilities, will be made available in the future.

For example, if the zenith optical depth is 0.1 nepers, then use the following parameters:

```
gaincal('data.ms', 'cal.G0', solint='inf', combine='', refant=11, opacity=0.1)
```

The calibration task in this example will apply an elevation-dependent opacity correction (scaled to 0.1 nepers at the zenith for all antennas for this example) calculated at each data sample before solving for gains on an “infinite” (up to scan boundaries) timescale.

If you do not have an externally supplied value for `opacity`, for example from a VLA tip procedure, then you should either use an average value for the telescope, or leave it at zero and let your gain calibration compensate as best it can (e.g. that your calibrator is at the same elevation as your target at approximately the same time. As noted above, there are no facilities yet to estimate this from the data (e.g. by plotting `Tsys` vs. elevation).

Below, we give instructions for determining `opacity` for VLA observations where tip-curve data is available. It is beyond the scope of this cookbook to provide information for other telescopes.

4.3.3.1 Determining opacity corrections for VLA data

For VLA data, zenith opacity can be measured at the frequency and during the time observations are made using a VLA tipping scan in the observe file. Historical tipping data are available at:

<http://www.vla.nrao.edu/astro/calib/tipper>

Choose a year, and click **Go** to get a list of all tipping scans that have been made for that year.

If a tipping scan was made for your observation, then select the appropriate file. Go to the bottom of the page and click on the button that says **Press here to continue..** The results of the tipping scan will be displayed. Go to the section called 'Overall Fit Summary' to find the fit quality and the fitted zenith opacity in percent. If the zenith opacity is reported as 6%, then the actual zenith optical depth value is `opacity=0.060` for `gaincal` and other calibration tasks.

If there were no tipping scans made for your observation, then look for others made in the same band around the same time and weather conditions. If nothing is available here, then at K and Q bands you might consider using an average value (e.g. 6% in reasonable weather). See the VLA memo

<http://www.vla.nrao.edu/memos/test/232/232.pdf>

for more on the atmospheric optical depth correction at the VLA, including plots of the seasonal variations.

4.3.4 Setting the Flux Density Scale using (`setjy`)

When solving for visibility-plane calibration, CASA calibration applications compare the observed `DATA` column with the `MODEL_DATA` column. The first time that an imaging or calibration task is executed for a given MS, the `MODEL_DATA` column is created and initialized with unit point source flux density visibilities (unpolarized) for all sources (e.g. `AMP=1`, `phase=0°`). The `setjy` task is then used to set the proper flux density for flux calibrators. For sources that are recognized flux calibrators (listed in Table 4.1), `setjy` will calculate the flux densities, Fourier transform them and write the results to the `MODEL_DATA` column. For the VLA, the default source models are customarily point sources defined by the Baars or Perley-Taylor flux density scales, or point sources of unit flux density if the flux density is unknown. The `MODEL_DATA` column can also be filled with a model

generated from an image of the source (e.g. the Fourier transform of an image generated after initial calibration of the data).

Table 4.1:

3C Name	B1950 Name	J2000 Name
3C286	1328+307	1331+305
3C48	0134+329	0137+331
3C147	0538+498	0542+498
3C138	0518+165	0521+166
–	1934-638	–
3C295	1409+524	1411+522

The inputs for `setjy` are:

```
# setjy :: Place flux density of sources in the measurement set:

vis           =      ''      # Name of input visibility file
field         =      ''      # Field name list or field ids list
spw           =      ''      # Spectral window identifier (list)
modimage      =      ''      # Model image name
fluxdensity   =      -1      # Specified flux density [I,Q,U,V]
standard     = 'Perley-Taylor 99' # Flux density standard
```

By default the `setjy` task will cycle through all fields and spectral windows, setting the flux density either to 1 Jy (unpolarized), or if the source is recognized as one of the calibrators in the above table, to the flux density (assumed unpolarized) appropriate to the observing frequency. For example, to run `setjy` on a measurement set called `data.ms`:

```
setjy(vis='data.ms') # This will set all fields and spectral windows
```

ALERT: At this time, all that `setjy` does is to fill the `MODEL_DATA` column of the MS with the Fourier transform of a source model. The `ft` task (§ 5.6) will do the same thing, although it does not offer the options for flux rescaling that `setjy` does. Note also that currently `setjy` will not transform a full-Stokes model image such that all polarizations are filled correct. You need to use `ft` for this.

To limit this operation to certain fields and spectral windows, use the `field` and/or `spw` parameters, which take the usual data selection strings (§ 2.3). For example, to set the flux density of the first field (all spectral windows)

```
setjy(vis='data.ms',field='0')
```

or to set the flux density of the second field in spectral window 17

```
setjy(vis='data.ms',field='1',spw='17')
```

The full-polarization flux density (I,Q,U,V) may also be explicitly provided:

```
setjy(vis='data.ms',
      field='1',spw='16',           # Run setjy on field id 1, spw id 17
      fluxdensity=[3.5,0.2,0.13,0.0]) # and set I,Q,U,V explicitly
```

Note: The `setjy` (or `ft`) operation is different than the antenna gain-elevation and atmospheric opacity Prior Calibrations (§ 4.3.2–4.3.3) in that it is applied to (and carried with) the MS itself, rather than via other tables or parameters to the subsequent tasks. It is more like the `Tsys` correction (§ 4.3.1) in this regard.

4.3.4.1 Using Calibration Models for Resolved Sources

If the flux density calibrator is resolved at the observing frequency, the point source model generated by `setjy` will not be appropriate. If available, a model image of the resolved source at the observing frequency may be used to generate the appropriate visibilities using the `modimage` parameter (or in older versions explicitly with the `ft` task). To use this, provide `modimage` with the path to the model image. Remember, if you just give the file name, it will assume that it is in the current working directory. Note also that `setjy` using a model image will only operate on that single source, thus you would run it multiple times (with different `field` settings) for different sources.

Otherwise, you may need to use the `uvrange` selection (§ 4.4.1.2) in the calibration solving tasks to exclude the baselines where the resolution effect is significant. There is not hard and fast rule for this, though you should consider this if your calibrator is shows a drop of more than 10% on the longest baselines (use `plotxy`, § 3.4.2, to look at this). You may need to do `antenna` selection also, if it is heavily resolved and there are few good baselines to the outer antennas. Note that `uvrange` may also be needed to exclude the short baselines on some calibrators that have extended flux not accounted for in the model. **Note:** the calibrator guides for the specific telescopes usually indicate appropriate min and max for `uvrange`. For example, see the *VLA Calibration Manual* at:

<http://www.vla.nrao.edu/astro/calib/manual/>

for details on the use of standard calibrators for the VLA.

Model images for some flux density calibrators are provided with CASA:

- Red Hat Linux RPMs 32bit (RHE4, Fedora 6): located in `/usr/lib/casapy/data/nrao/VLA/CalModels`
- Red Hat Linux RPMs 64bit (RHE4, Fedora 6): located in `/usr/lib64/casapy/data/nrao/VLA/CalModels`
- MAC OSX .dmg: located in `/Applications/CASA.app/Contents/Resources/casa-data/nrao/VLA/CalModels`
- NRAO-AOC casapy-test: `/home/casa/data/nrao/VLA/CalModels`

e.g., these are found in the `data/nrao/VLA/CalModels` sub-directory of the CASA installation. For example, just point to the repository copy, e.g.

```
modimage = '/usr/lib/casapy/data/nrao/VLA/CalModels/3C48_C.im'
```

or if you like, you can copy the ones you wish to use to your working directory.

The models available are:

```
3C138_L.im          3C286_L.im  3C48_L.im
3C138_C.im 3C147_C.im 3C286_C.im 3C48_C.im
3C138_X.im 3C147_X.im 3C286_X.im 3C48_X.im
3C138_U.im 3C147_U.im 3C286_U.im 3C48_U.im
3C138_K.im 3C147_K.im 3C286_K.im 3C48_K.im
3C138_Q.im 3C147_Q.im 3C286_Q.im 3C48_Q.im
```

These are all un-reconvolved images of AIPS CC lists, properly scaled to the Perley-Taylor 1999 flux density for the frequencies at which they were observed.

It is important that the model image *not* be one convolved with a finite beam; it must have units of Jy/pixel (not Jy/beam).

Note that `setjy` will rescale the flux in the models for known sources (e.g. those in Table 4.1) to match those it would have calculated. It will thus extrapolated the flux out of the frequency band of the model image to whatever spectral windows in the MS are specified (but will use the structure of the source in the model image).

ALERT: The reference position in the `modimage` is currently used by `setjy` when it does the Fourier transform, thus differences from the positions for the calibrator in the MS will show up as phase gradients in the uv-plane. If your model image position is significantly different but you don't want this to affect your calibration, then you can doctor either the image header using `imhead` (§ 6.2) or in the MS (using the `ms` tool) as appropriate. In an upcoming release we will put in a toggle to use or ignore the position of the `modimage`. Note that this will not affect the flux scaling (only put in erroneous model phases); in any event small position differences, such as those arising by changing epoch from B1950 to J2000 using `regridimage` (§ 6.9), will be inconsequential to the calibration.

This illustrates the use of `uvrange` for a slightly resolved calibrator:

```
# Import the data
importvla(archivefiles='AS776_A031015.xp2', vis='ngc7538_XBAND.ms',
          freqtol=10000000.0, bandname='X')

# Flag the ACs
flagautocorr('ngc7538_XBAND.ms')

# METHOD 1: Use point source model for 3C48, plus uvrange in solve

# Use point source model for 3C48
setjy(vis='ngc7538_XBAND.ms',field='0');
```



```

# Limit 3C48 (fieldid=0) solutions to uvrage = 0-40 klamba
gaincal(vis='ngc7538_XBAND.ms', caltable='cal.G', field='0',
        solint=60.0, refant='10', selectdata=True, uvrage='0~40klamba',
        append=False, gaincurve=False, opacity=0.0)

# Append phase-calibrator's solutions (no uvrage) to the same table
gaincal(vis='ngc7538_XBAND.ms', caltable='cal.G', field='2',
        solint=60.0, refant='10', selectdata=True, uvrage='',
        append=True, gaincurve=False, opacity=0.0)

# Fluxscale
fluxscale(vis='ngc7538_XBAND.ms', caltable='cal.G', reference=['0137+331'],
          transfer=['2230+697'], fluxtable='cal.Gflx', append=False)

```

while the following illustrates the use of of a model:

```

# METHOD 2: use a resolved model copied from the data respository
#   for 3C48, and no uvrage
# (NB: detailed freq-dep flux scaling TBD)

# Copy the model image 3C48_X.im to the working directory first!

setjy(vis='ngc7538_XBAND.ms', field='0', modimage='3C48_X.im')

# Solutions on both calibrators with no uvrage
gaincal(vis='ngc7538_XBAND.ms', caltable='cal.G2', field='0,2',
        solint=60.0, refant='10',
        append=False, gaincurve=False, opacity=0.0)

# Fluxscale
fluxscale(vis='ngc7538_XBAND.ms', caltable='cal.G2', reference=['0137+331'],
          transfer=['2230+697'], fluxtable='cal.G2flx', append=False)

# Both methods give 2230 flux densities ~0.7 Jy, in good agreement with
#   AIPS

```

4.3.5 Correction for delay and antenna position offsets using gencal

The `gencal` task provides a means of specifying antenna-based calibration values manually. The values are put in designated tables and can be applied to the data on-the-fly in solving tasks and using `applycal`.

The `gencal` task has the inputs:

```

# gencal :: Externally specify calibration solutions of varous types
vis           =      ''      # Name of input visibility file
caltable      =      ''      # The input/output calibration table
caltype       =      ''      # The calibration type

```

```

spw           =      ''           # Calibration spw(s) selection
antenna       =      ''           # Calibraiton antenna(s) selection
pol           =      ''           # Calibration polarizations(s) selection
parameter     =      []           # The calibration values
async        =      False        # If true the taskname must be started using gencal(...)

```

Current antenna-based gencal options (caltype) are:

- 'amp' — amplitude correction
- 'ph' — phase correction
- 'sbd' — single-band delay (phase-frequency slope for each spw)
- 'mbd' — multi-band delay (phase-frequency slope over all spw)
- 'antpos' — ITRF antenna position corrections
- 'antposvla' — VLA-centric antenna position corrections

The calibration specifications cannot be time-variable in the present implementation. Calibration values can be assigned to each `spw`, `antenna` and `pol` selection, where applicable. The list of calibration values specified in `parameter` must conform to the range of spectral windows, antennas, and polarizations specified in `spw`, `antenna` and `pol`, with the values specified in order of the specified polarizations (fastest), antennas, and spectral windows (slowest). If any of `spw`, `antenna`, or `pol` are left unspecified (empty strings), the values specified in `parameter` will be assumed applicable to all values of the unspecified data axes. The output caltable will otherwise assume nominal calibration values for unspecified spectral windows, antennas, and polarizations. Note that antenna position corrections formally do not have spectral-window or polarization dependence; such specifications should not be used with 'antpos'.

The same caltable can be specified for multiple runs of gencal, in which case the specified parameters will be incorporated cumulatively. E.g., amplitude parameters (`calttype='amp'`) multiply and phase-like parameters ('ph', 'sbd', 'mbd', 'antpos') add. Parameters for 'amp' and 'ph' corrections can be incorporated into the same caltable (in separate runs), but each of the other types require their own unique caltable. A mechanism for specifying manual corrections via a text file will be provided in the future.

Two kinds of delay corrections are supported. For `calttype='sbd'`, the specified delays (in nanoseconds) will be applied locally to each spectral window, referring the derived phase corrections to each spectral window's reference frequency (where the phase correction will be zero). The phases in each spectral window will nominally be flattened, but any phase offsets between spectral windows will remain. (These can be corrected using `calttype='phase'`, or via ordinary spectral-window-dependent phase calibration.) For `calttype='mbd'`, the derived phase corrections are referred to zero frequency. This causes a correction that is coherent over many spectral windows. If the data are already coherent over many spectral windows and share a common multi-band delay (e.g., EVLA data), `calttype='mbd'` corrections will maintain this coherence and flatten the frequency-dependent phase. Using `calttype='sbd'` in this instance will introduce phase offsets among spectral windows that reflect the multi-band delay.

For antenna position corrections (`caltype='antpos'`), the antenna position offsets are specified in the ITRF frame. For VLA position corrections in the VLA-centric frame, use `caltype='antposvla'`, and `gencal` will rotate them to ITRF before storing them in the output caltable.

The sign and scale convention for `gencal` corrections (indeed for all CASA caltables) is such that the specified parameters (and as stored in caltables) are the factors that *corrupt* ideal data to yield the observed data. Thus, when applied to correct the data, their effective inverse will automatically be taken. I.e., amplitude factors will be divided into the data on correction. Phase-like parameters adopt the convention that the complex factor for the second antenna in the baseline is conjugated, and then both antenna factors are divided into the data on correction. (These conventions differ from AIPS in that `multiplying` correction factors are stored in AIPS calibration tables; however, the phase convention ends up being the same since AIPS conjugates the complex factor for the *first* antenna in the baseline.)

The following series of examples illustrate the use of `gencal`.

For the dataset `'data.ms'`, the following sequence of `gencal` runs introduces, into a single caltable (`'test.G'`), (1) an antenna-based amplitude scale correction of 3.0 for all polarizations, antennas, and spectral windows, (2) phase corrections for all spectral windows and polarizations of 45 and 120 degrees to antennas EA03 and EA04, respectively, (3) phase corrections for all spectral windows of 63 and -34 in R (only) for antennas EA05 and EA06, respectively, and (4) phase corrections for all spectral windows of 14, -23, -130, and 145 degrees for antenna/polarizations EA09/R, EA09/L, EA10/R, and EA10/L, respectively:

```
gencal(vis='data.ms',caltable='test.G',caltype='amp', \
      spw='',antenna='',pol='', \
      parameter=[3])
```

```
gencal(vis='data.ms',caltable='test.G',caltype='ph', \
      spw='',antenna='EA03,EA04',pol='', \
      parameter=[45,120])
```

```
gencal(vis='data.ms',caltable='test.G',caltype='ph', \
      spw='',antenna='EA05,EA06',pol='R', \
      parameter=[63,-34])
```

```
gencal(vis='data.ms',caltable='test.G',caltype='ph', \
      spw='',antenna='EA09,EA10',pol='R,L', \
      parameter=[14,-23,-130,145])
```

In the following example, delay corrections in both polarizations will be adjusted for antenna EA09 by 14 nsec in spw 2 and -130 nsec in spw 3, and for antenna EA10 by -23 nsec in spw 2 and 145 nsec in spw 3:

```
gencal(vis='test.ms',caltable='test.sbd',caltype='sbd', \
      spw='2,3',antenna='EA09,EA10',pol='', \
      parameter=[14,-23,-130,145])
```

In the following example, antenna position corrections in meters (in ITRF) for antenna EA09 (dBx=0.01, dBy=0.02, dBz=0.03) and for antenna EA10 (dBx=-0.03, dBy=-0.01, dBz=-0.02) are introduced. Note that three parameters are required for each antenna.

```
gencal(vis='test.ms', caltable='test.antpos', caltype='antpos', \
       antenna='EA09,EA10', \
       parameter=[0.01,0.02,0.03, -0.03,-0.01,-0.02])
```

In the following example, antenna position corrections (in the traditional VLA-centric frame) will be introduced in meters for antenna EA09 (dBx=0.01, dBy=0.02, dBz=0.03) and for antenna EA10 (dBx=-0.03, dBy=-0.01, dBz=-0.02) These offsets will be rotated to the ITRF frame before storing them in the caltable.

```
gencal(vis='test.ms', caltable='test.antposvla', caltype='antposvla', \
       antenna='EA09,EA10', \
       parameter=[0.01,0.02,0.03, -0.03,-0.01,-0.02])
```

4.3.6 Other *a priori* Calibrations and Corrections

Other *a priori* calibrations will be added to the `calibrator` (`cb`) tool in the near future. These will include system temperature normalization (amplitude) corrections, tropospheric phase corrections derived from Water Vapor Radiometry (WVR) measurements, instrumental line-length corrections, etc. Where appropriate, solving capabilities for these effects will also be added.

4.4 Solving for Calibration — Bandpass, Gain, Polarization

These tasks actually solve for the unknown calibration parameters, placing the results in a calibration table. They take as input an MS, and a number of parameters that specify any prior calibration or previous calibration tables to pre-apply before computing the solution. These are placed in the proper sequence of the Measurement Equation automatically.

We first discuss the parameters that are in common between many of the calibration tasks. Then we describe each solver in turn.

4.4.1 Common Calibration Solver Parameters

There are a number of parameters that are in common between the calibration “solver” tasks. These also appear in some of the other calibration manipulation and application tasks.

4.4.1.1 Parameters for Specification : vis and caltable

The input measurement set and output table are controlled by the following parameters:

```
vis          =          '' # Name of input visibility file
caltable     =          '' # Name of output calibration table
```

The MS name is input in `vis`. If it is highlighted red in the inputs (§ 1.3.5.4) then it does not exist, and the task will not execute. Check the name and path in this case.

The output table name is placed in `caltable`. Be sure to give a unique name to the output table, or be careful. If the table exists, then what happens next will depend on the task and the values of other parameters (e.g. § 4.4.1.6). The task may not execute giving a warning that the table already exists, or will go ahead and overwrite the solutions in that table, or append them. Be careful.

4.4.1.2 Selection: field, spw, and selectdata

Selection is controlled by the parameters:

```
field        =          '' # field names or index of calibrators: ''==>all
spw          =          '' # spectral window:channels: ''==>all
selectdata   =      False # Other data selection parameters
```

Field and spectral window selection are so often used, that we have made these standard parameters `field` and `spw` respectively.

The `selectdata` parameter expands as usual, uncovering other selection sub-parameters:

```
selectdata   =          True # Other data selection parameters
  timerange  =          '' # time range: ''==>all
  uvrange    =          '' # uv range''==>all
  antenna    =          '' # antenna/baselines: ''==>all
  scan       =          '' # scan numbers: Not yet implemented
  msselect   =          '' # Optional data selection (Specialized. but see help)
```

Note that if `selectdata=False` these parameters are not used when the task is executed, even if set underneath.

The most common `selectdata` parameter to use is `uvrange`, which can be used to exclude longer baselines if the calibrator is resolved, or short baselines of the calibrator contains extended flux not accounted for in the model (e.g. § 4.3.4.1).

See § 2.3 for more on the selection parameters.

4.4.1.3 Prior Calibration and Correction: `parang`, `gaincurve` and `opacity`

These parameters control the on-the-fly application of various calibration or effect-based corrections prior to the solving process.

The `parang` parameter turns on the application of the antenna-based parallactic angle correction ('P') in the measurement equation. This is necessary for polarization calibration and imaging, or for cases where the parallactic angles are different for geographically spaced antennas (e.g. VLBI). For dealing with only the parallel-hand corrections (e.g. RR, LL, XX, YY) for a co-located array (e.g. the VLA or ALMA), you can set `parang=False` and save some computational effort. Otherwise, set `parang=True` to apply this correction.

There are two control parameters for applying Prior Calibration:

```
gaincurve    =      False # Apply VLA antenna gain curve correction
opacity      =      0.0  # Opacity correction to apply (nepers)
```

See § 4.3 for more on **Prior Calibration**.

4.4.1.4 Previous Calibration: `gaintable`, `gainfield`, `interp` and `spwmap`

Calibration tables that have already been determined can also be applied before solving for the new table:

```
gaintable    =      '' # Prior gain calibration table(s) to apply
gainfield    =      '' # Field selection on prior gaintable(s)
interp       =      '' # Interpolation mode (in time) for prior gaintable(s)
spwmap       =      [] # Spectral window mapping for each gaintable (see help)
```

This is controlled by the `gaintable` parameter, which takes a string or list of strings giving one or more calibration tables to pre-apply. For example,

```
gaintable = ['ngc5921.bcal', 'ngc5921.gcal']
```

specifies two tables, in this case bandpass and gain calibration tables respectively.

The other parameters key off `gaintable`, taking single values or lists, with an entry for each table in `gaintable`. The order is given by that in `gaintable`.

The `gainfield` parameter specifies which fields from the respective `gaintable` to use to apply. This is a list, with each entry a string or list of strings. The default '' for an entry means to use all in that table. For example,

```
gaintable = ['ngc5921.bcal', 'ngc5921.gcal']
gainfield = [ '1331+305', ['1331+305', '1445+099'] ]
```

or using indices

```
gainfield = [ '0', ['0','1'] ]
```

to specify the field '1331+305' from the table 'ngc5921.bcal' and fields '1331+305' and '1445+099' from the second table 'ngc5921.gcal'. We could also have wildcarded the selection, e.g.

```
gainfield = [ '0', '*' ]
```

taking all fields from the second table. And of course we could have used the default

```
gainfield = [ '0', '' ]
```

or even

```
gainfield = [ '0' ]
```

which is to take all.

The `interp` parameter chooses the interpolation scheme to be used when pre-applying the solution in the tables. This interpolation is (currently) only in time. The choices are currently 'nearest', 'linear', and 'aipslin':

- 'nearest' just picks the entry nearest in time to the visibility in question;
- 'linear' interpolation calibrates each datum with calibration phases and amplitudes linearly interpolated from neighboring time values. In the case of phase, this mode will assume that phase jumps greater than 180° between neighboring points indicate a cycle slip, and the interpolated value will follow this change in cycle accordingly;
- 'aipslin' emulates the classic AIPS interpolation mode with linearly interpolated amplitudes and phases derived from interpolation of the complex calibration values. While this method avoids having to track cycle slips (which is unstable for solutions with very low SNR), it will yield a phase interpolation which becomes increasingly non-linear as the spanned phase difference increases. The non-linearity mimics the behavior of `interp='nearest'` as the spanned phase difference approaches 180° (the phase of the interpolated complex calibration value initially changes very slowly, then rapidly jumps to the second value at the midpoint of the interval).

If the uncalibrated phase is changing rapidly, a 'nearest' interpolation is not desirable. Usually, `interp='linear'` is the best choice. For example,

```
interp = [ 'nearest', 'linear' ]
```

uses nearest “interpolation” on the first table, and linear on the second.

The `spwmap` parameter sets the spectral window combinations to form for the `gaintable(s)`. This is a list, or a list of lists, of integers giving the `spw` IDs to map. There is one list for each table in `gaintable`, with an entry for each ID in the MS. For example,

```
spwmap=[0,0,1,1]          # apply from spw=0 to 0,1 and 1 to 2,3
```

for an MS with `spw=0,1,2,3`. For multiple gaintable, use lists of lists, e.g.

```
spwmap=[ [0,0,1,1], [0,1,0,1] ] # 2nd table spw=0 to 0,2 and 1 to 1,3
```

4.4.1.5 Solving: `solint`, `combine`, `preavg`, `refant`, `minblperant`, `minsnr`

The parameters controlling common aspects of the solution are:

```
solint      =      'inf'   # Solution interval: egs. 'inf', '60s' (see help)
combine     =      ''     # Data axes which to combine for solve (scan, spw, and/or field)
preavg      =      -1.0   # Pre-averaging interval (sec) (rarely needed)
refant      =      ''     # Reference antenna name:''=no explicit reference
minblperant =      4     # Minimum baselines _per antenna_ required for solve
minsnr      =      0.0   # Reject solutions below this SNR: 0==>no rejection
```

The solution interval is given by `solint`. If given a number without a unit, this is in seconds. The special values `'inf'` and `-1` specify an “infinite” solution interval encompassing the entire dataset, while `'int'` or zero specify a solution every integration. You can use time quanta in the string, e.g. `solint='1m'` and `solint='60s'` both specify solution intervals of one minute. Note that `solint` interacts with `combine` to determine whether the solutions cross scan or field boundaries.

The parameter controlling the scope of the solution is `combine`. For the default `combine=''` solutions will break at scan, field, and spw boundaries. Specification of any of these in `combine` will extend the solutions over the boundaries (up to the `solint`). For example, `combine='spw'` will combine spectral windows together for solving, while `combine='scan'` will cross scans. Thus, to do scan-based solutions (single solution for each scan), set

```
solint = 'inf'
combine = ''
```

while

```
solint = 'inf'
combine = 'scan'
```

will make a single solution for the entire dataset (for a given field and spw). You can specify multiple choices for combination:

```
combine = 'scan,spw'
```

for example.

The reference antenna is specified by the `refant` parameter. This useful to “lock” the solutions with time, effectively rotating (after solving) the phase of the gain solution for the reference antenna to be zero (the exact effect depends on the type of solution). You can also run without a reference

antenna, but in this case the solutions will float with time, with a phase that rotates around with the relative weights of the antennas in the solution (its more or less like setting the weighted sum of the antenna phases to zero). It is usually prudent to select an antenna in the center of the array that is known to be particularly stable, as any gain jumps or wanders in the `refant` will be transferred to the other antenna solutions.

Although rarely needed, setting a `preavg` time will let you average data over periods shorter than the solution interval first before solving on longer timescales.

The minimum signal-to-noise ratio allowed for an acceptable solution is specified in the `minsnr` parameter.

The `minblperant` parameter sets the minimum number of baselines to other antennas that must be preset for a given antenna to get a solution.

4.4.1.6 Action: `append` and `solnorm`

The following parameters control some things that happen after solutions are obtained:

```
solnorm      =      False  #   Normalize solution amplitudes post-solve.
append       =      False  #   Append solutions to (existing) table.  False will overwrite.
```

The `solnorm` parameter toggles on the option to normalize the solution amplitudes after the solutions are obtained. The exact effect of this depends upon the type of solution. Not all tasks include this parameter.

One should be aware when using `solnorm` that if this is done in the last stage of a chain of calibration, then the part of the calibration that is “normalized” away will be lost. It is best to use this in early stages (for example in a first bandpass calibration) so that later stages (such as final gain calibration) can absorb the lost normalization scaling. It is not strictly necessary to use `solnorm=True` at all, but is sometimes helpful if you want to have a normalized bandpass for example.

The `append` parameter, if set to `True`, will append the solutions from this run to existing solutions in `caltable`. Of course, this only matters if the table already exists. If `append=False` and `caltable` exists, it will overwrite.

4.4.2 Spectral Bandpass Calibration (`bandpass`)

For channelized data, it is often desirable to solve for the gain variations in frequency as well as in time. Variation in frequency arises as a result of non-uniform filter passbands or other dispersive effects in signal transmission. It is usually the case that these frequency-dependent effects vary on timescales much longer than the time-dependent effects handled by the gain types 'G' and 'T'. Thus, it makes sense to solve for them as a separate term: 'B', using the `bandpass` task.

The inputs to `bandpass` are:

```

# bandpass :: Calculate a bandpass solution

vis          =      '' # Nome of input visibility file
caltable     =      '' # Name of output gain calibration table
field        =      '' # Select field using field id(s) or field name(s)
spw          =      '' # Select spectral window/channels
selectdata   =      False # Other data selection parameters
solint       =      'inf' # Solution interval
combine      =      'scan' # Data axes which to combine for solve (scan, spw, and/or field)
refant       =      '' # Reference antenna name
minblperant  =      4 # Minimum baselines _per antenna_ required for solve
solnorm      =      False # Normalize average solution amplitudes to 1.0
bandtype     =      'B' # Type of bandpass solution (B or BPOLY)
  fillgaps   =      0 # Fill flagged solution channels by interpolation
append       =      False # Append solutions to the (existing) table
gaintable    =      [''] # Gain calibration table(s) to apply on the fly
gainfield    =      [''] # Select a subset of calibrators from gaintable(s)
interp       =      [''] # Interpolation mode (in time) to use for each gaintable
spwmap       =      [] # Spectral windows combinations to form for gaintables(s)
gaincurve    =      False # Apply internal VLA antenna gain curve correction
opacity      =      0.0 # Opacity correction to apply (nepers)
parang       =      False # Apply parallactic angle correction
async        =      False # if True run in the background, prompt is freed

```

Many of these parameters are in common with the other calibration tasks and are described above in § 4.4.1.

The `bandtype` parameter selects the type of solution used for the bandpass. The choices are 'B' and 'BPOLY'. The former solves for a complex gain in each channel in the selected part of the MS. See § 4.4.2.2 for more on 'B'. The latter uses a polynomial as a function of channel to fit the bandpass, and expands further to reveal a number of sub-parameters See § 4.4.2.3 for more on 'BPOLY'.

It is usually best to solve for the bandpass in channel data before solving for the gain as a function of time. However, if the gains of the bandpass calibrator observations are fluctuating over the timerange of those observations, then it can be helpful to first solve for the gains of that source with `gaincal`, and input these to `bandpass` via `gaintable`. See more below on this strategy.

We now describe the issue of bandpass normalization, followed by a description of the options `bandtype='B'` and `bandtype='BPOLY'`.

4.4.2.1 Bandpass Normalization

The `solnorm` parameter (§ 4.4.1.6) deserves more explanation in the context of the bandpass. Most users are used to seeing a normalized bandpass, where the vector sum of the antenna-based channel gains sums to unity amplitude and zero phase. The toggle `solnorm=True` allows this. However, the parts of the bandpass solution normalized away will be still left in the data, and thus you should not use `solnorm=True` if the `bandpass` calibration is the end of your calibration sequence (e.g. you

have already done all the gain calibration you want to). Note that setting `solnorm=True` will NOT rescale any previous calibration tables that the user may have supplied in `gaintable`.

You can safely use `solnorm=True` if you do the bandpass first (perhaps after a throw-away initial gain calibration) as we suggest above in § 4.2, as later gain calibration stages will deal with this remaining calibration term. This does have the benefit of isolating the overall (channel independent) gains to the following `gaincal` stage. It is also recommended for the case where you have multiple scans on possibly different bandpass calibrators. It may also be preferred when applying the bandpass before doing `gaincal` and then `fluxscale` (§ 4.4.4), as significant variation of bandpass among antennas could otherwise enter the gain solution and make (probably subtle) adjustments to the flux scale.

We finally note that `solnorm=False` at the bandpass step in the calibration chain will in the end produce the correct results. It only means that there will be a part of what we usually think of the gain calibration inside the bandpass solution, particularly if `bandpass` is run as the first step.

4.4.2.2 B solutions

Calibration type 'B' differs from 'G' only in that it is determined for each channel in each spectral window. It is possible to solve for it as a function of time, but it is most efficient to keep the 'B' solving timescale as long as possible, and use 'G' or 'T' for rapid frequency-independent time-scale variations.

The 'B' solutions are limited by the signal-to-noise ratio available per channel, which may be quite small. It is therefore important that the data be coherent over the time-range of the 'B' solutions. As a result, 'B' solutions are almost always preceded by an initial 'G' or 'T' solve using `gaincal` (§ 4.4.3). In turn, if the 'B' solution improves the frequency domain coherence significantly, a 'G' or 'T' solution following it will be better than the original.

For example, to solve for a 'B' bandpass using a single short scan on the calibrator, then

```
default('bandpass')

vis = 'n5921.ms'
caltable = 'n5921.bcal'
gaintable = ''           # No gain tables yet
gainfield = ''
interp = ''
field = '0'             # Calibrator 1331+305 = 3C286 (FIELD_ID 0)
spw = ''               # all channels
selectdata = False     # No other selection
gaincurve = False      # No gaincurve at L-band
opacity = 0.0          # No troposphere
bandtype = 'B'         # standard time-binned B (rather than BPOLY)
solint = 'inf'         # set solution interval arbitrarily long
refant = '15'          # ref antenna 15 (=VLA:N2) (ID 14)

bandpass()
```

On the other hand, we might have a number of scans on the bandpass calibrator spread over time, but we want a single bandpass solution. In this case, we could solve for and then pre-apply an initial gain calibration, and let the bandpass solution cross scans:

```
gaintable = 'n5921.init.gcal'    # Our previously determined G table
gainfield = '0'
interp = 'linear'               # Do linear interpolation
solint = 'inf'                  # One interval over dataset
combine = 'scan'                # Solution crosses scans
```

Note that we obtained a bandpass solution for all channels in the MS. If explicit channel selection is desired, for example some channels are useless and can be avoided entirely (e.g. edge channels or those dominated by Gibbs ringing), then `spw` can be set to select only these channels, e.g.

```
spw = '0:4~59'                  # channels 4-59 of spw 0
```

This is not so critical for 'B' solutions as for 'BPOLY', as each channel is solved for independently, and poor solutions can be dropped.

If you have multiple time solutions, then these will be applied using whatever interpolation scheme is specified in later tasks.

The `combine` parameter (§ 4.4.1.5) can be used to combine data across spectral windows, scans, and fields.

4.4.2.3 BPOLY solutions

For some observations, it may be the case that the SNR per channel is insufficient to obtain a usable per-channel 'B' solution. In this case it is desirable to solve instead for a best-fit functional form for each antenna using the `bandtype='BPOLY'` solver. The 'BPOLY' solver naturally enough fits (Chebychev) polynomials to the amplitude and phase of the calibrator visibilities as a function of frequency. Unlike ordinary 'B', a single common 'BPOLY' solution will be determined for all spectral windows specified (or implicit) in the selection. As such, it is usually most meaningful to select individual spectral windows for 'BPOLY' solves, unless groups of adjacent spectral windows are known *a priori* to share a single continuous bandpass response over their combined frequency range (e.g., PdBI data).

The 'BPOLY' solver requires a number of unique sub-parameters:

```
bandtype      = 'BPOLY'    # Type of bandpass solution (B or BPOLY)
degamp        = 3          # Polynomial degree for BPOLY amplitude solution
degphase      = 3          # Polynomial degree for BPOLY phase solution
visnorm       = False     # Normalize data prior to BPOLY solution
maskcenter    = 0          # Number of channels in BPOLY to avoid in center of band
maskededge    = 0          # Percent of channels in BPOLY to avoid at each band edge
```

The `degamp` and `degphase` parameters indicate the polynomial degree desired for the amplitude and phase solutions. The `maskcenter` parameter is used to indicate the number of channels in the center of the band to avoid passing to the solution (e.g., to avoid Gibbs ringing in central channels for PdBI data). The `maskedge` drops beginning and end channels. The `visnorm` parameter turns on normalization before the solution is obtained (rather than after for `solnorm`).

The `combine` parameter (§ 4.4.1.5) can be used to combine data across spectral windows, scans, and fields.

Note that `bandpass` will allow you to use multiple `fields`, and can determine a single solution for all specified fields using `combine='field'`. If you want to use more than one field in the solution it is prudent to use an initial `gaincal` using proper flux densities for all sources (not just 1Jy) and use this table as an input to `bandpass` because in general the phase towards two (widely separated) sources will not be sufficiently similar to combine them, and you want the same amplitude scale. If you do not include amplitude in the initial `gaincal`, you probably want to set `visnorm=True` also to take out the amplitude normalization change. Note also in the case of multiple `fields`, that the 'BPOLY' solution will be labeled with the field ID of the first `field` used in the 'BPOLY' solution, so if for example you point `plotcal` at the name or ID of one of the other fields used in the solution, `plotcal` does not plot.

For example, to solve for a 'BPOLY' (5th order in amplitude, 7th order in phase), using data from field 2, with G corrections pre-applied:

```
bandpass(vis='data.ms',          # input data set
         caltable='cal.BPOLY',   #
         spw='0:2~56',          # Use channels 3-57 (avoid end channels)
         field='0',              # Select bandpass calibrator (field 0)
         bandtype='BPOLY',      # Select bandpass polynomials
         degamp=5,               # 5th order amp
         degphase=7,             # 7th order phase
         gaintable='cal.G',      # Pre-apply gain solutions derived previously
         refant='14')            #
```

4.4.3 Complex Gain Calibration (`gaincal`)

The fundamental calibration to be done on your interferometer data is to calibrate the antenna-based gains as a function of time in the various frequency channels and polarizations. Some of these calibrations are known beforehand (“a priori”) and others must be determined from observations of calibrators, or from observations of the target itself (“self-calibration”).

It is best to have removed a (slowly-varying) “bandpass” from the frequency channels by solving for the bandpass (see above). Thus, the `bandpass` calibration table would be input to `gaincal` via the `gaintable` parameter (see below).

The `gaincal` task has the following inputs:

```
# gaincal :: Determine temporal gains from calibrator observations:
```

```

vis          =      '' # Nome of input visibility file
caltable     =      '' # Name of output gain calibration table
field       =      '' # Select field using field id(s) or field name(s)
spw         =      '' # Select spectral window/channels
selectdata  =      False # Other data selection parameters
solint      =      'inf' # Solution interval (see help)
combine     =      '' # Data axes which to combine for solve (scan, spw, and/or field)
preavg      =      -1.0 # Pre-averaging interval (sec)
refant      =      '' # Reference antenna name
minblperant =      4 # Minimum baselines _per antenna_ required for solve
minsnr      =      0.0 # Reject solutions below this SNR
solnorm     =      False # Normalize average solution amplitudes to 1.0 (G, T only)
gaintype    =      'G' # Type of gain solution (G, T, or GSPLINE)
calmode     =      'ap' # Type of solution" ('ap', 'p', 'a')
append      =      False # Append solutions to the (existing) table
gaintable   =      [''] # Gain calibration table(s) to apply on the fly
gainfield   =      [''] # Select a subset of calibrators from gaintable(s)
interp      =      [''] # Interpolation mode (in time) to use for each gaintable
spwmap      =      [] # Spectral windows combinations to form for gaintables(s)
gaincurve   =      False # Apply internal VLA antenna gain curve correction
opacity     =      0.0 # Opacity correction to apply (nepers)
parang      =      False # Apply parallactic angle correction
async       =      False

```

Data selection is done through the standard `field`, `spw` and `selectdata` expandable sub-parameters (see § 2.3). The bulk of the other parameters are the standard solver parameters. See § 4.4.1 above for a description of these.

The `gaintype` parameter selects the type of gain solution to compute. The choices are 'T', 'G', and 'GSPLINE'. The 'G' and 'T' options solve for independent complex gains in each solution interval (classic AIPS style), with 'T' enforcing a single polarization-independent gain for each co-polar correlation (e.g. RR and LL, or XX and YY) and 'G' having independent gains for these. See § 4.4.3.1 for a more detailed description of 'G' solutions, and § 4.4.3.2 for more on 'T'. The 'GSPLINE' fits cubic splines to the gain as a function of time. See § 4.4.3.3 for more on this option.

4.4.3.1 Polarization-dependent Gain (G)

Systematic time-dependent complex gain errors are almost always the dominant calibration effect, and a solution for them is almost always necessary before proceeding with any other calibration. Traditionally, this calibration type has been a catch-all for a variety of similar effects, including: the relative amplitude and phase gain for each antenna, phase and amplitude drifts in the electronics of each antenna, amplitude response as a function of elevation (gain curve), and tropospheric amplitude and phase effects. In CASA, it is possible to handle many of these effects separately, as available information and circumstances warrant, but it is still possible to solve for the net effect using calibration type G.

Generally speaking, type G can represent any per-spectral window multiplicative polarization- and time-dependent complex gain effect downstream of the polarizers. (Polarization *independent* effects

upstream of the polarizers may also be treated with G.) Multi-channel data (per spectral window) will be averaged in frequency before solving (use calibration type B to solve for frequency-dependent effects within each spectral window).

To solve for G on, say, fields 1 & 2, on a 90s timescale, and apply, e.g., gain curve corrections:

```
gaincal('data.ms',
        caltable='cal.G',      # Write solutions to disk file 'cal.G'
        field='0,1',          # Restrict field selection
        solint=90.0,          # Solve for phase and amp on a 90s timescale
        gaincurve=True        # Note: gaincurve=False by default
        refant=3)              #

plotcal('cal.G','amp')       # Inspect solutions
```

These G solution will be referenced to antenna 4. Choose a well-behaved antenna that is located near the center of the array for the reference antenna. For non-polarization datasets, reference antennas need not be specified although you can if you want. If no reference antenna is specified, an effective phase reference that is an average over the data will be calculated and used. For data that requires polarization calibration, you must choose a reference antenna that has a constant phase difference between the right and left polarizations (e.g. no phase jumps or drifts). If no reference antenna (or a poor one) is specified, the phase reference may have jumps in the R-L phase, and the resulting polarization angle response will vary during the observation, thus corrupting the polarization imaging.

To apply this solution to the calibrators and the target source (field 2, say):

```
applycal('data.ms',
         field='0,1,2',      # Restrict field selection (cals + src)
         opacity=0.0,        # Don't apply opacity correction
         gaintable='cal.G')  # Apply G solutions and correct data
plotxy('data.ms',xaxis='channel',datacolumn='data',subplot=211)
plotxy('data.ms',xaxis='channel',datacolumn='corrected',subplot=212)
```

The calibrated data is written to the CORRECTED_DATA column, with `calwt=True` by default.

Alert: *Current* (as of June 2010) EVLA data has no weights to the data. To avoid trouble, `calwt=False` should be set for those data sets. Older VLA data should still be calibrated with `calwt=True`.

4.4.3.2 Polarization-independent Gain (T)

At high frequencies, it is often the case that the most rapid time-dependent gain errors are introduced by the troposphere, and are polarization-independent. It is therefore unnecessary to solve for separate time-dependent solutions for both polarizations, as is the case for 'G'. Calibration type 'T' is available to calibrate such tropospheric effects, differing from 'G' only in that a single common solution for both polarizations is determined. In cases where only one polarization is observed, type 'T' is adequate to describe the time-dependent complex multiplicative gain calibration.

In the following example, we assume we have a 'G' solution obtained on a longish timescale (longer than a few minutes, say), and we want a residual 'T' solution to track the polarization-independent variations on a very short timescale:

```
gaincal('data.ms',          # Visibility dataset
        caltable='cal.T',   # Specify output table name
        gaintype='T',       # Solve for T
        field='0,1',        # Restrict data selection to calibrators
        solint=3.0,         # Obtain solutions on a 3s timescale
        gaintable='cal120.G') # Pre-apply prior G solution
```

For dual-polarization observations, it will always be necessary to obtain a 'G' solution to account for differences and drifts between the polarizations (which traverse different electronics), but solutions for rapidly varying polarization-independent effects such as those introduced by the troposphere will be optimized by using 'T'. Note that 'T' can be used in this way for self-calibration purposes, too.

4.4.3.3 GSPLINE solutions

At high radio frequencies, where tropospheric phase fluctuates rapidly, it is often the case that there is insufficient signal-to-noise ratio to obtain robust 'G' or 'T' solutions on timescales short enough to track the variation. In this case it is desirable to solve for a best-fit functional form for each antenna using the 'GSPLINE' solver. This fits a time-series of cubic B-splines to the phase and/or amplitude of the calibrator visibilities.

The `combine` parameter (§ 4.4.1.5) can be used to combine data across spectral windows, scans, and fields. Note that if you want to use `combine='field'`, then all fields used to obtain a 'GSPLINE' amplitude solution must have models with accurate relative flux densities. Use of incorrect relative flux densities will introduce spurious variations in the 'GSPLINE' amplitude solution.

The 'GSPLINE' solver requires a number of unique additional parameters, compared to ordinary 'G' and 'T' solving. The sub-parameters are:

```
gaintype      = 'GSPLINE' # Type of solution (G, T, or GSPLINE)
  splinetime  =   3600.0 # Spline (smooth) timescale (sec), default=1 hours
  npointaver  =         3 # Points to average for phase wrap (okay)
  phasewrap   =        180 # Wrap phase when greater than this (okay)
```

The duration of each spline segment is controlled by `splinetime`. The actual `splinetime` will be adjusted such that an integral number of equal-length spline segments will fit within the overall range of data.

Phase splines require that cycle ambiguities be resolved prior to the fit; this operation is controlled by `npointaver` and `phasewrap`. The `npointaver` parameter controls how many contiguous points in the time-series are used to predict the cycle ambiguity of the next point in the time-series, and `phasewrap` sets the threshold phase jump (in degrees) that would indicate a cycle slip. Large values

of `npointaver` improve the SNR of the cycle estimate, but tend to frustrate ambiguity detection if the phase rates are large. The `phaseswrap` parameter may be adjusted to influence when cycles are detected. Generally speaking, large values ($> 180^\circ$) are useful when SNR is high and phase rates are low. Smaller values for `phaseswrap` can force cycle slip detection when low SNR conspires to obscure the jump, but the algorithm becomes significantly less robust. More robust algorithms for phase-tracking are under development (including fringe-fitting).

For example, to solve for 'GSPLINE' phase and amplitudes, with splines of duration 600 seconds,

```
gaincal('data.ms',
        caltable='cal.spline.ap',
        gaintype='GSPLINE'      # Solve for GSPLINE
        calmode='ap'           # Solve for amp & phase
        field='0,1',           # Restrict data selection to calibrators
        splinetime=600.)      # Set spline timescale to 10min
```

ALERT: The 'GSPLINE' solutions can not yet be used in `fluxscale`. You should do at least some 'G' amplitude solutions to establish the flux scale, then do 'GSPLINE' in phase before or after to fix up the short timescale variations. Note that the "phase tracking" algorithm in 'GSPLINE' needs some improvement.

4.4.4 Establishing the Flux Density Scale (`fluxscale`)

The 'G' or 'T' solutions obtained from calibrators for which the flux density was unknown and assumed to be 1 Jansky are correct in a time- and antenna- relative sense, but are mis-scaled by a factor equal to the inverse of the square root of the true flux density. This scaling can be corrected by enforcing the constraint that mean gain amplitudes determined from calibrators of unknown flux density should be the same as determined from those with known flux densities. The `fluxscale` task exists for this purpose.

The inputs for `fluxscale` are:

```
# fluxscale :: Bootstrap the flux density scale from standard calibrators
vis          =      '' # Name of input visibility file
caltable     =      '' # Name of input calibration table
fluxtable    =      '' # Name of output, flux-scaled calibration table
reference    =      '' # Reference field name(s) (transfer flux scale FROM)
transfer     =      '' # Transfer field name(s) (transfer flux scale TO), '' -> all
append       =      False # Append solutions?
refspwmap    =      [-1] # Scale across spectral window boundaries. See help fluxscale
async        =      False # If true the taskname must be started using fluxscale(...)
```

Before running `fluxscale`, one must have first run `setjy` for the reference sources and run a `gaincal` on both `reference` and `transfer` fields. After running `fluxscale` the output `fluxtable` `caltable` will have been scaled such that the correct scaling will be applied to the `transfer` sources.

For example, given a 'G' table, e.g. 'cal.G', containing solutions for a flux density calibrator (in this case '3C286') and for one or more gain calibrator sources with unknown flux densities (in this example '0234+285' and '0323+022'):

```
fluxscale(vis='data.ms',
          caltable='cal.G',           # Select input table
          fluxtable='cal.Gflx',      # Write scaled solutions to cal.Gflx
          reference='3C286',         # 3C286 = flux calibrator
          transfer='0234+258, 0323+022') # Select calibrators to scale
```

The output table, 'cal.Gflx', contains solutions that are properly scaled for all calibrators.

Note that the assertion that the gain solutions are independent of the calibrator includes the assumption that the gain amplitudes are strictly not systematically time dependent. While synthesis antennas are designed as much as possible to achieve this goal, in practice, a number of effects conspire to frustrate it. When relevant, it is advisable to pre-apply `gaincurve` and `opacity` corrections when solving for the 'G' solutions that will be flux-scaled (see § 4.3 and § 4.4.1.3). When the 'G' solutions are essentially constant for each calibrator separately, the `fluxscale` operation is likely to be robust.

The `fluxscale` task can be executed on either 'G' or 'T' solutions, but it should only be used on one of these types if solutions exist for both and one was solved relative to the other (use `fluxscale` only on the first of the two).

ALERT: The 'GSPLINE' option is not yet supported in `fluxscale` (see § 4.4.3.3).

If the `reference` and `transfer` fields were observed in different spectral windows, the `refspwmap` parameter may be used to achieve the scaling calculation across spectral window boundaries.

The `refspwmap` parameter functions similarly to the standard `spwmap` parameter (§ 4.4.1.4), and takes a list of indices indicating the spectral window mapping for the reference fields, such that `refspwmap[i]=j` means that reference field amplitudes from spectral window `j` will be used for spectral window `i`.

Note: You should be careful when you have a dataset with spectral windows with different bandwidths, and you have observed the calibrators differently in the different `spw`. The flux-scaling will probably be different in windows with different bandwidths.

For example,

```
fluxscale(vis='data.ms',
          caltable='cal.G',           # Select input table
          fluxtable='cal.Gflx',      # Write scaled solutions to cal.Gflx
          reference='3C286',         # 3C286 = flux calibrator
          transfer='0234+258,0323+022' # Select calibrators to scale
          refspwmap=[0,0,0])         # Use spwid 0 scaling for spwids 1 & 2
```

will use `spw=0` to scale the others, while in

```
fluxscale(vis='data.ms',
          caltable='cal.G',           # Select input table
          fluxtable='cal.Gflx',      # Write scaled solutions to cal.Gflx
          reference='3C286',         # 3C286 = flux calibrator,
          transfer='0234+285, 0323+022', # select calibrators to scale,
          refspwmap=[0,0,1,1])       # select spwids for scaling,
```

the reference amplitudes from spectral window 0 will be used for spectral windows 0 and 1 and reference amplitudes from spectral window 2 will be used for spectral windows 2 and 3.

4.4.4.1 Using Resolved Calibrators

If the flux density calibrator is resolved, the assumption that it is a point source will cause solutions on outlying antennas to be biased in amplitude. In turn, the `fluxscale` step will be biased on these antennas as well. In general, it is best to use model for the calibrator, but if such a model is not available, it is important to limit the solution on the flux density calibrator to only the subset of antennas that have baselines short enough that the point-source assumption is valid. This can be done by using `antenna` and `uvrange` selection when solving for the flux density calibrator. For example, if antennas 1 through 8 are the antennas among which the baselines are short enough that the point-source assumption is valid, and we want to be sure to limit the solutions to the use of baselines shorter than 15000 wavelengths, then we can assemble properly scaled solutions for the other calibrator as follows (note: specifying both an antenna and a `uvrange` constraint prevents inclusion of antennas with only a small number of baselines within the specified `uvrange` from being included in the solution; such antennas will have poorly constrained solutions):

As an example, we first solve for gain solutions for the flux density calibrator (3C286 observed in field 0) using a subset of antennas

```
gaincal(vis='data.ms',
        caltable='cal.G',      # write solutions to cal.G
        field='0'             # Select the flux density calibrator
        selectdata=True,     # Expand other selectors
        antenna='0~7',       # antennas 0-7,
        uvrange='0~15klambda', # limit uvrange to 0-15klambda
        solint=90)           # on 90s timescales, write solutions
                             # to table called cal.G
```

Now solve for other calibrator (0234+285 in field 1) using all antennas (implicitly) and append these solutions to the same table

```
gaincal(vis='data.ms',
        caltable='cal.G',      # write solutions to cal.G
        field='1',
        solint=90,
        append=T)             # Set up to write to the same table
```

Finally, run `fluxscale` to adjust scaling

```
fluxscale(vis='data.ms',
          caltable='cal.G',    # Input table with unscaled cal solutions
          fluxtable='cal.Gflx', # Write scaled solutions to cal.Gflx
          reference='3C286',   # Use 3c286 as ref with limited uvrange
          transfer='0234+285') # Transfer scaling to 0234+285
```

The `fluxscale` calculation will be performed using only the antennas common to both fields, but the result will be applied to all antennas on the transfer field. Note that one can nominally get by only with the `uvrange` selection, but you may find that you get strange effects from some antennas only having visibilities to a subset of the baselines and thus causing problems in the solving.

4.4.5 Instrumental Polarization Calibration (D,X)

The inputs to `polcal` are:

```
# polcal :: Determine instrumental polarization from calibrator observations
vis      =      '' # Nome of input visibility file
caltable =      '' # Name of output gain calibration table
field    =      '' # Select field using field id(s) or field name(s)
spw      =      '' # Select spectral window/channels
selectdata = False # Other data selection parameters
solint   =      'inf' # Solution interval
combine  =      'scan' # Data axes which to combine for solve (scan, spw, and/or field)
preavg   =      300.0 # Pre-averaging interval (sec)
refant   =      '' # Reference antenna name
minblperant = 4 # Minimum baselines _per antenna_ required for solve
minsnr   =      0.0 # Reject solutions below this SNR
poltype  =      'D+QU' # Type of instrumental polarization solution (see help)
append   =      False # Append solutions to the (existing) table
gaintable = [''] # Gain calibration table(s) to apply
gainfield = [''] # Select a subset of calibrators from gaintable(s)
interp   =      [''] # Interpolation mode (in time) to use for each gaintable
spwmap   =      [] # Spectral windows combinations to form for gaintables(s)
gaincurve = False # Apply internal VLA antenna gain curve correction
opacity  =      0.0 # Opacity correction to apply (nepers)
async    =      False
```

The `polcal` task uses many of the standard calibration parameters as described above in § 4.4.1.

The key parameter controlling `polcal` is `poltype`. The choices are:

- 'D' — Solve for instrumental polarization (leakage D-terms), using the transform of an IQU model in `MODEL_DATA`; requires no parallactic angle coverage, but if the source polarization is non-zero, the gain calibration must have the correct R-L phase registration. (Note: this is unlikely, so just use 'D+X' to let the position angle registration float.) This will produce a calibration table of type **D**.
- 'D+X' — Solve for instrumental polarization D-terms and the polarization position angle correction, using the transform of an IQU model in `MODEL_DATA`; this mode requires at least 2 distinct parallactic angles to separate the net instrumental polarization and the PA. This will produce a calibration table of type 'D'. **ALERT:** no table of type 'X' will be produced, so you must follow this by a run of `polcal` with `polmode='X'` (see below).

'D+QU' — Solve for instrumental polarization and source $Q + iU$; requires at least 3 distinct parallactic angles to separate the net instrumental polarization from the source Q and U . Effectively sets the polarization PA to the value if the R-L phase difference were 0° . This will produce a calibration table of type 'D'.

'X' — Solve only for the position angle correction; best to use this after getting the D-terms from one of the above modes. Requires the observation of a calibrator with known $Q + iU$ (or at least known U/Q). This will produce a calibration table of type 'X'.

There are channelized solution modes for the above options. For example, substitute 'Df' for 'D' in the 'D*' modes described above to get a channelized D-term solution. **ALERT:** 'X' solutions are currently always frequency-independent.

ALERT: `polcal` will obtain a separate D-term solution for each `field` supplied to it. This limitation will be relaxed in the future, enabling more sensitive solutions, as well as flexibilities like solving for 'D+X' using a single scan each of two or more position angle calibrators.

4.4.5.1 Heuristics and Strategies for Polarization Calibration

Fundamentally, with good ordinary gain (and bandpass, if relevant) calibration already in hand, good polarization calibration must deliver both the instrumental polarization and position angle calibration. An unpolarized source can deliver only the first of these, but does not require parallactic angle coverage. A polarized source can only deliver the position angle calibration also if its polarization is known a priori. Sources that are polarized, but with unknown polarization, must always be observed with sufficient parallactic angle coverage, where "sufficient" is determined by SNR and the details of the solving mode.

These principles are stated assuming the instrumental polarization solution is solved using the "linear approximation" where cross-terms in more than a single product of the instrumental or source polarizations are ignored in the Measurement Equation (see § E). A general non-linearized solution, with sufficient SNR, may enable some relaxation of the requirements indicated here.

For instrumental polarization calibration, there are 3 types of calibrator choice:

CASA Polarization Calibration Modes

Cal Polarization	Parallactic Angles	MODEL_DATA	polmode	Result
unpolarized	any	set $Q = U = 0$	'D' or 'Df'	D-terms only
known non-zero	2+ scans	set Q, U	'D+X' or 'Df+X'	D-terms and PA
unknown	3+ scans	ignored	'D+QU' or 'Df+QU'	D-terms and source

Note that the parallactic angle ranges spanned by the scans in the modes that require this should be large enough to give good separation between the components of the solution. In practice, 60° is a good target.

Each of these solutions should be followed with a 'X' solution on a source with known polarization position angle (and correct $Q + iU$ in MODEL_DATA). **ALERT:** `polmode='D+X'` will soon deliver this automatically.

The `polcal` task will solve for the 'D' or 'X' terms using the model visibilities that are in the `MODEL_DATA` column of the MS. Calibration of the parallel hands must have already been carried out using `gaincal` and/or `bandpass` in order to align the phases over time and frequency. This calibration need not have been applied and can be supplied through the `gaintable` parameters, but any cal-tables to be used in `polcal` must agree (e.g. have been derived from) the data in the `DATA` column and the model visibilities in the `MODEL_DATA` column of the MS. Thus, for example, one would not use the cal-table produced by `fluxscale` as the rescaled amplitudes would no longer agree with the contents of `MODEL_DATA`.

Be careful when using resolved calibrators for polarization calibration. A particular problem is if the structure in Q and U is offset from that in I. Use of a point model, or a resolved model for I but point models for Q and U, can lead to errors in the 'X' calibration. Use of a `uvrange` will help here. The use of a full-Stokes model with the correct polarization is the only way to ensure a correct calibration if these offsets are large.

4.4.5.2 A Polarization Calibration Example

In the following example, we do a standard 'D+QU' solution on the bright source BLLac (2202+422) which has been tracked through a range in parallactic angle:

```
default('polcal')
vis           = 'polcal_20080224.cband.all.ms'
caltable     = 'polcal_20080224.cband.all.pcal'
field        = '2202+422'
spw          = ''
solint       = 'inf'
combine      = 'scan'
preavg       = 300.0
refant       = 'VA15'
minsnr       = 3
poltype      = 'D+QU'
gaintable    = 'polcal_20080224.cband.all.gcal'
gainfield    = ['']
polcal()
```

This assumes `setjy` and `gaincal` have already been run. Note that the original gain-calibration table is used in `gaintable` so that what is in the `MODEL_DATA` column is in agreement with what is in the `gaintable`, rather than using the table resulting from `fluxscale`.

A bit later on, we need to set the R-L phase using a scan on 3C48 (0137+331):

```
default('polcal')
vis           = 'polcal_20080224.cband.all.ms'
caltable     = 'polcal_20080224.cband.all.polx'
field        = '0137+331'
refant       = 'VA15'
minsnr       = 3
poltype      = 'X'
```

```
gaintable = ['polcal_20080224.cb主and.all.gcal', 'polcal_20080224.cb主and.all.pcal']
polcal()
```

If, on the other hand, we had a scan on an unpolarized bright source, for example 3C84 (0319+415), we could use this to calibrate the leakages:

```
default('polcal')
vis = 'polcal_20080224.cb主and.all.ms'
caltable = 'polcal_20080224.cb主and.all_3c84.pcal'
field = '0319+415'
refant = 'VA15'
poltype = 'D'
gaintable = 'polcal_20080224.cb主and.all.gcal'
polcal()
```

We would then do the 'X' calibration as before (but using this D-table in `gaintable`).

A full processing example for continuum polarimetry can be found in § F.2.

4.4.6 Baseline-based Calibration (`blcal`)

You can use the `blcal` task to solve for baseline-dependent (non-closing) errors. **WARNING:** this is in general a very dangerous thing to do, since baseline-dependent errors once introduced are difficult to remove. You must be sure you have an excellent model for the source (better than the magnitude of the baseline-dependent errors).

The inputs are:

```
# blcal :: Calculate a baseline-based calibration solution (gain or bandpass)
vis = '' # Nome of input visibility file
caltable = '' # Name of output gain calibration table
field = '' # Select field using field id(s) or field name(s)
spw = '' # Select spectral window/channels
selectdata = False # Other data selection parameters
solint = 'inf' # Solution interval
combine = '' # Data axes which to combine for solve (scan, spw, and/or field)
freqdep = False # Solve for frequency dependent solutions
calmode = 'ap' # Type of solution" ('ap', 'p', 'a')
solnorm = False # Normalize average solution amplitudes to 1.0
gaintable = [''] # Gain calibration table(s) to apply on the fly
gainfield = [''] # Select a subset of calibrators from gaintable(s)
interp = [''] # Interpolation mode (in time) to use for each gaintable
spwmap = [] # Spectral windows combinations to form for gaintables(s)
gaincurve = False # Apply internal VLA antenna gain curve correction
opacity = 0.0 # Opacity correction to apply (nepers)
parang = False # Apply parallactic angle correction
async = False # If true the taskname must be started using blcal(...)
```

The `freqdep` parameter controls whether `blcal` solves for “gain” (`freqdep=False`) or “bandpass” (`freqdep=True`) style calibration.

Other parameters are the same as in other calibration tasks. These common calibration parameters are described in § 4.4.1.

4.5 Plotting and Manipulating Calibration Tables

At some point, the user should examine (plotting or listing) the calibration solutions. Calibration tables can also be manipulated in various ways, such as by interpolating between times (and sources), smoothing of solutions, and accumulating various separate calibrations into a single table.

4.5.1 Plotting Calibration Solutions (`plotcal`)

The `plotcal` task is available for examining solutions of all of the basic solvable types (G, T, B, D, M, MF, K). The inputs are:

```
# plotcal :: An all-purpose plotter for calibration results:

caltable      =      '' # Name of input calibration table
xaxis         =      '' # Value to plot along x axis (time,chan,amp,phase,real,imag,snr)
yaxis         =      '' # Value to plot along y axis (amp,phase,real,imag,snr)
poln          =      '' # Polarization to plot (RL,R,L,XY,X,Y,/)
field         =      '' # Field names or index: ''=all, '3C286,P1321*', '0~3'
antenna       =      '' # Antenna selection. E.g., antenna='3~5'
spw           =      '' # Spectral window: ''=all, '0,1' means spw 0 and 1
timerange     =      '' # Time selection ''=all
subplot       =      111 # Panel number on display screen (yxn)
overplot      =      False # Overplot solutions on existing display
clearpanel    =      'Auto' # Specify if old plots are cleared or not
iteration     =      '' # Iterate on antenna,time,spw,field
plotrange     =      [] # plot axes ranges: [xmin,xmax,ymin,ymax]
showflags     =      False # If true, show flags
plotsymbol    =      '.' # pylab plot symbol
plotcolor     =      'blue' # initial plotting color
markersize   =      5.0 # size of plot symbols
fontsize      =      10.0 # size of label font
showgui       =      True # Show plot on gui
figfile       =      '' # ''= no plot hardcopy, otherwise supply name
```

ALERT: Currently, `plotcal` needs to know the MS from which `caltable` was derived to get indexing information. It does this using the name stored inside the table, which does not include the full path, but assumes the MS is in the `cwd`. Thus if you are using a MS in a directory other than the current one, it will not find it. You need to change directories using `cd` in IPython (or `os.chdir()` inside a script) to the MS location.

The controls for the `plotcal` window are the same as for `plotxy` (see § 3.4.2.1).

The `xaxis` and `yaxis` plot options available are:

- `'amp'` — amplitude,
- `'phase'` — phase,
- `'real'` — the real part,
- `'imag'` — the imaginary part,
- `'snr'` — the signal-to-noise ratio,

of the calibration solutions that are in the `caltable`. The `xaxis` choices also include `'time'` and `'channel'` which will be used as the sensible defaults (if `xaxis=''`) for gain and bandpass solutions respectively.

The `poln` parameter determines what polarization or combination of polarization is being plotted. The `poln='RL'` plots both R and L polarizations on the same plot. The respective XY options do equivalent things. The `poln='/'` option plots amplitude ratios or phase differences between whatever polarizations are in the MS (R and L. or X and Y).

The `field`, `spw`, and `antenna` selection parameters are available to obtain plots of subsets of solutions. The syntax for selection is given in § 2.3.

The `subplot` parameter is particularly helpful in making multi-panel plots. The format is `subplot=yxn` where `yxn` is an integer with digit `y` representing the number of plots in the y-axis, digit `x` the number of panels along the x-axis, and digit `n` giving the location of the plot in the panel array (where `n = 1, ..., xy`, in order upper left to right, then down). See § 3.4.2.8 for more details on this option.

The `iteration` parameter allows you to select an identifier to iterate over when producing multi-panel plots. The choices for `iteration` are: `'antenna'`, `'time'`, `'spw'`, `'field'`. For example, if per-antenna solution plots are desired, use `iteration='antenna'`. You can then use `subplot` to specify the number of plots to appear on each page. In this case, set the `n` to 1 for `subplot=yxn`. Use the **Next** button on the plotcal window to advance to the next set of plots. Note that if there is more than one timestamp in a 'B' table, the user will be queried to interactively advance the plot to each timestamp, or if `multiplot=True`, the antennas plots will be cycled through for each timestamp in turn. Note that `iteration` can take more than one iteration choice (as a single string containing a comma-separated list of the options). **ALERT:** the iteration order is fixed (independent of the order specified in the `iteration` string), for example:

```
iteration = 'antenna, time, field'
iteration = 'time, antenna, field'
```

will both iterate over each field (fastest) then time (next) and antenna (slowest). The order is:

```
iteration = 'antenna, time, field, spw'
```

from the slowest (outer loop) to fastest (inner loop).

The `markersize` and `fontsize` parameters are especially helpful in making the dot and label sizes appropriate for the plot being made. The screen shots in this section used this feature to make the plots more readable in the cookbook. Adjusting the `fontsize` can be tricky on multi-panel plots, as the labels can run together if too large. You can also help yourself by manually resizing the Plotter window to get better aspect ratios on the plots.

ALERT: Unfortunately, `plotcal` has many of the same problems that `plotxy` does, as they use similar code underneath. An overhaul is underway, so stay tuned.

4.5.1.1 Examples for `plotcal`

For example, to plot amplitude or phase as a function of time for 'G' solutions (after rescaling by `fluxscale` for the NGC5921 “demo” data (see Appendix F.1),

```
default('plotcal')
fontsize = 14.0    # Make labels larger
markersize = 10.0 # Make dots bigger

caltable = 'ngc5921.usecase.fluxscale'
yaxis = 'amp'
subplot = 211
plotcal()

yaxis = 'phase'
subplot = 212
plotcal()
```

The results are shown in Figure 4.3. This makes use of the `subplot` option to make multi-panel displays.

Similarly, to plot amplitude or phase as a function of channel for 'B' solutions for NGC5921:

```
default('plotcal')
fontsize = 14.0    # Make labels larger
markersize = 10.0 # Make dots bigger

caltable = 'ngc5921.usecase.bcal'
antenna = '1'
yaxis = 'amp'
subplot = 311
plotcal()

yaxis = 'phase'
subplot = 312
plotcal()

yaxis = 'snr'
```

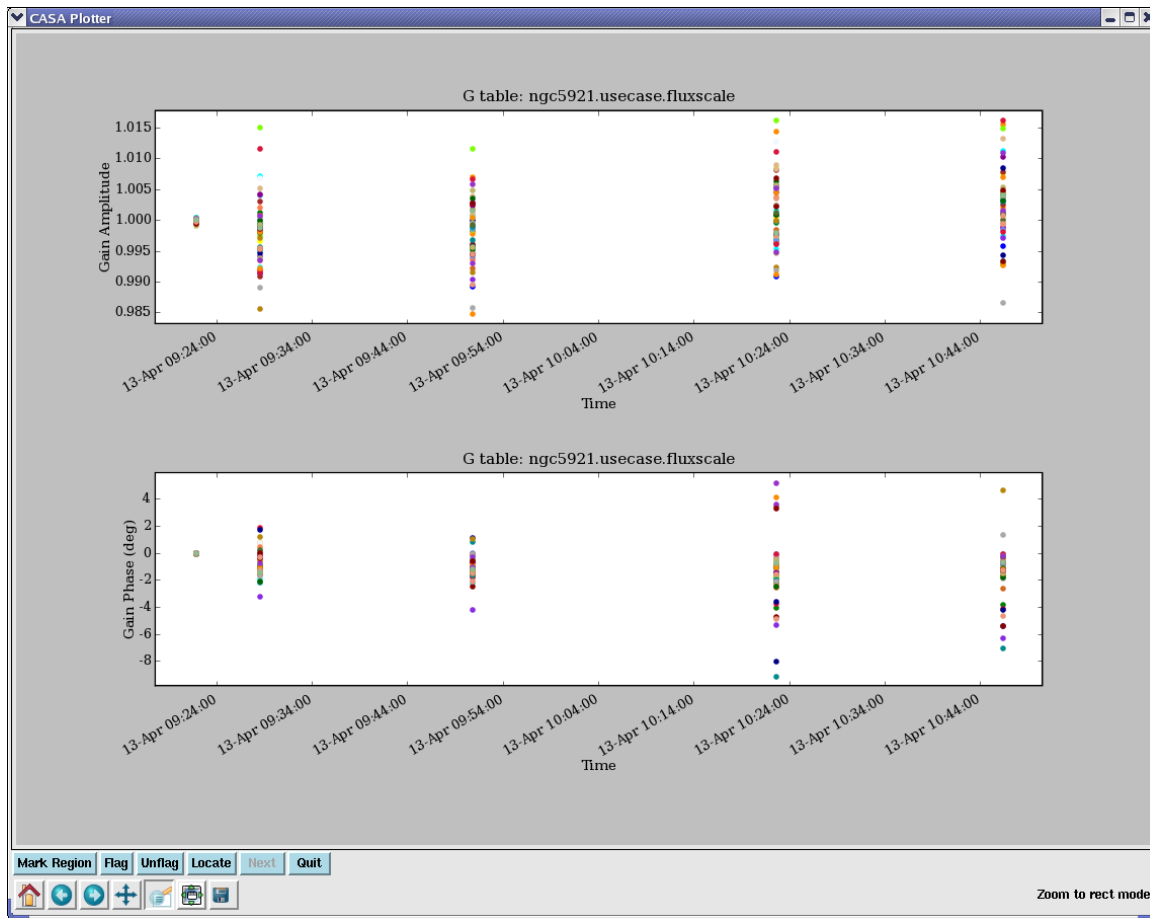


Figure 4.3: Display of the amplitude (upper) and phase (lower) gain solutions for all antennas and polarizations in the `ngc5921` post-fluxscale table.

```
subplot = 313
plotcal()
```

The results are shown in Figure 4.4. This stacks three panels with amplitude, phase, and signal-to-noise ratio. We have picked `antenna='1'` to show.

For example, to show 6 plots per page of 'B' amplitudes on a 3×2 grid:

```
default('plotcal')
fontsize = 12.0    # Make labels just large enough
markersize = 10.0 # Make dots bigger

caltable = 'ngc5921.usecase.bcal'
yaxis = 'amp'
subplot = 231
```

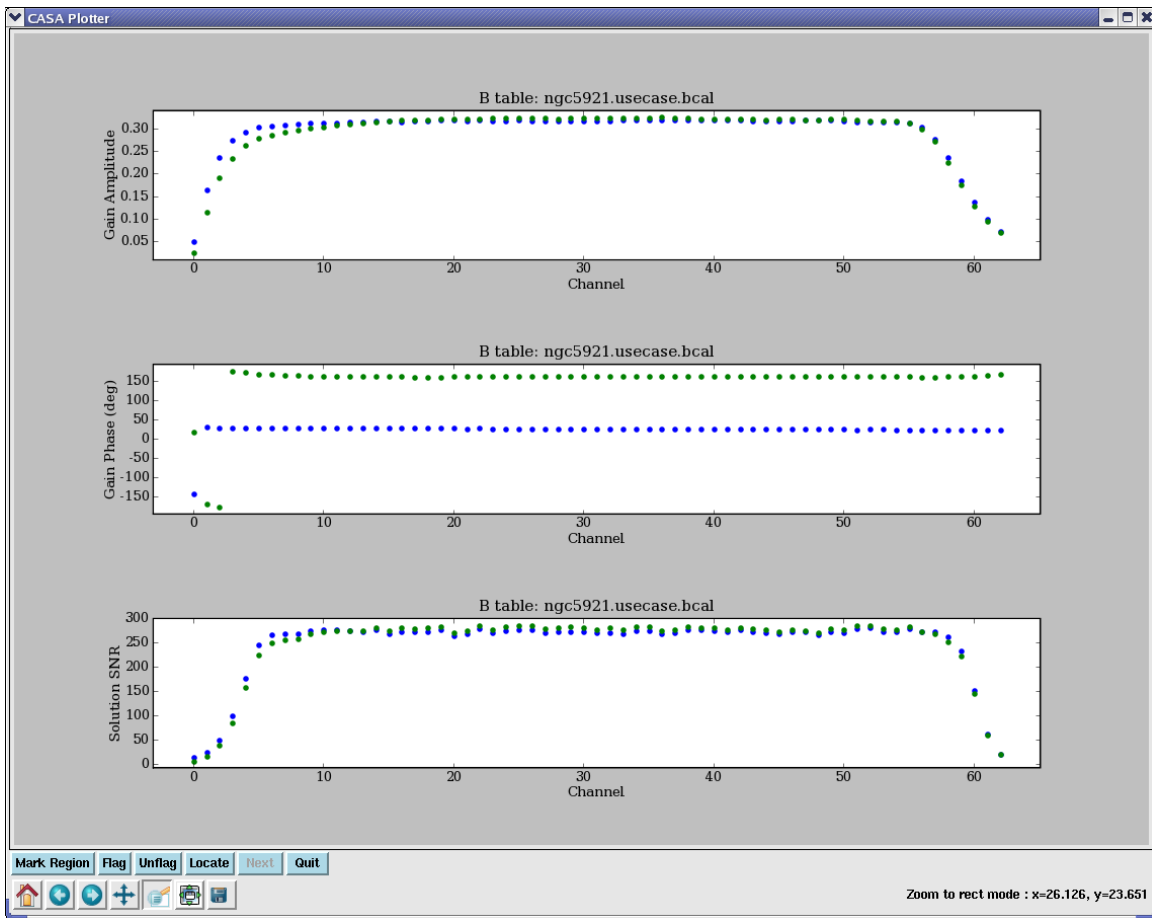


Figure 4.4: Display of the amplitude (upper), phase (middle), and signal-to-noise ratio (lower) of the bandpass 'B' solutions for `antenna='0'` and both polarizations for `ngc5921`. Note the falloff of the SNR at the band edges in the lower panel.

```
iteration = 'antenna'
```

```
plotcal()
```

See Figure 4.5 for this example. This uses the `iteration` parameter.

4.5.2 Listing calibration solutions with (`listcal`)

The `listcal` task will list the solutions in a specified calibration table.

The inputs are:

```
# listcal :: List data set summary in the logger:
```

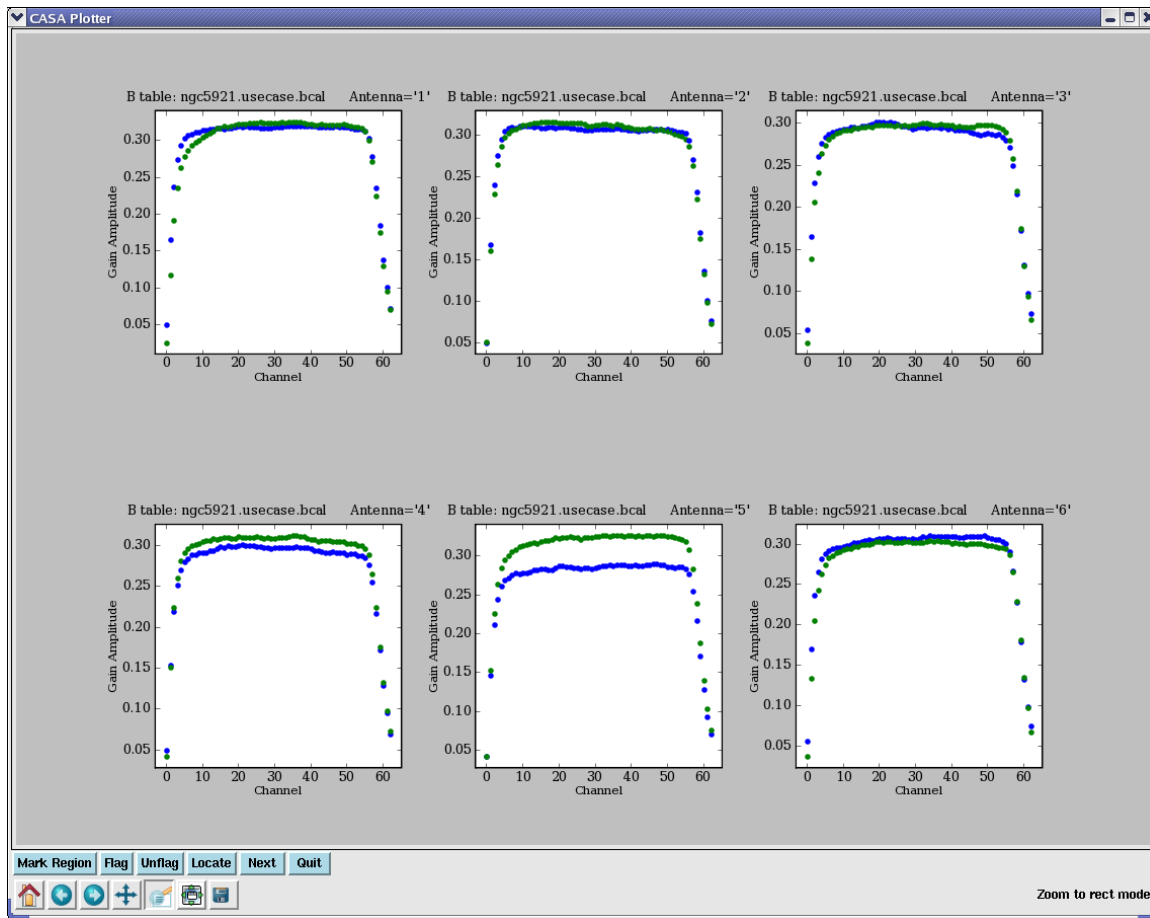


Figure 4.5: Display of the amplitude of the `bandpass 'B'` solutions. Iteration over antennas was turned on using `iteration='antenna'`. The first page is shown. The user would use the `Next` button to advance to the next set of antennas.

```
vis          =          '' # Name of input visibility file (MS)
caltable     =          '' # Input calibration table to list
field        =          '' # Select data based on field name or index
antenna      =          '' # Select data based on antenna name or index
spw          =          '' # Spectral window, channel to list
listfile     =          '' # Disk file to write, else to terminal
pagerows    =          50 # Rows listed per page
async       =          False
```

An example listing is:

```
Listing CalTable: jupiter6cm.usecase.split.ms.smoothcal2 (G Jones)
```

SpwId = 0, channel = 0.

Time	Field	Ant	:	Amp	Phase	Amp	Phase
1999/04/16/14:10:43.5	'JUPITER'	'1'	:	1.016	-11.5	1.016	-9.2
		'2'	:	1.013	-5.3	0.993	-3.1
		'3'	:	0.993	-0.8	0.990	-5.1
		'4'	:	0.997	-10.7	0.999	-8.3
		'5'	:	0.985	-2.7	0.988	-4.0
		'6'	:	1.005	-8.4	1.009	-5.3
		'7'	:	0.894	-8.7	0.897	-6.8
		'8'	:	1.001	-0.1	0.992	-0.7
		'9'	:	0.989	-12.4	0.992	-13.5
		'10'	:	1.000F	-4.2F	1.000F	-3.2F
		'11'	:	0.896	-0.0	0.890	-0.0
		'12'	:	0.996	-10.6	0.996	-4.2
		'13'	:	1.009	-8.4	1.011	-6.1
		'14'	:	0.993	-17.6	0.994	-16.1
		'15'	:	1.002	-0.8	1.002	-1.1
		'16'	:	1.010	-9.9	1.012	-8.6
		'17'	:	1.014	-8.0	1.017	-7.1
		'18'	:	0.998	-3.0	1.005	-1.0
		'19'	:	0.997	-39.1	0.994	-38.9
		'20'	:	0.984	-5.7	0.986	3.0
		'21'	:	1.000F	-4.2F	1.000F	-3.2F
		'22'	:	1.003	-11.8	1.004	-10.4
		'23'	:	1.007	-13.8	1.009	-11.7
		'24'	:	1.000F	-4.2F	1.000F	-3.2F
		'25'	:	1.000F	-4.2F	1.000F	-3.2F
		'26'	:	0.992	3.7	1.000	-0.2
		'27'	:	0.994	-5.6	0.991	-4.3
		'28'	:	0.993	-10.7	0.997	-3.8

4.5.3 Calibration table statistics (calstat)

The calstat task will print the statistics of solutions in a specified calibration table.

The inputs are:

```
# calstat :: Displays statistical information on a calibration table
caltable      =      ''      # Name of input calibration table
axis          =      'amp'   # Which values to use
datacolumn   =      'gain'   # Which data column to use

useflags     =      True    # Take flagging into account? (not implemented)
async       =      False   # If true the taskname must be started using calstat(...)
```

For example:

```
CASA <3>: calstat('ngc5921.demo.gcal',axis='amp',datacolumn='gain')
```

```
Out [3]:
{'GAIN': {'max': 1.6031942367553711,
          'mean': 1.4448433067117419,
          'medabsdevmed': 0.0086394548416137695,
          'median': 1.5732669830322266,
          'min': 0.99916577339172363,
          'npts': 280.0,
          'quartile': 0.020265340805053711,
          'rms': 1.4650156497955322,
          'stddev': 0.24271160321065546,
          'sum': 404.55612587928772,
          'sumsq': 600.95579999685287,
          'var': 0.058908922333086665}}}
```

```
CASA <4>: calstat('ngc5921.demo.gcal',axis='phase',datacolumn='gain')
```

```
Out [4]:
{'GAIN': {'max': 0.091214209794998169,
          'mean': -0.015221830284565011,
          'medabsdevmed': 0.012778861448168755,
          'median': -0.012778861448168755,
          'min': -0.15903720259666443,
          'npts': 280.0,
          'quartile': 0.02537553571164608,
          'rms': 0.031241731718182564,
          'stddev': 0.027331476552707856,
          'sum': -4.2621124796782031,
          'sumsq': 0.27329283416317834,
          'var': 0.00074700961055121926}}}
```

The statistics can be captured as return variables from the task:

```
CASA <7>: mystat = calstat('ngc5921.demo.gcal',axis='amp',datacolumn='gain')
```

```
CASA <8>: print 'Gain Amp = ',mystat['GAIN']['mean'],'+/-',mystat['GAIN']['stddev']
Gain Amp = 1.44484330671 +/- 0.242711603211
```

ALERT: This task is still under development and currently offers no selection (e.g. by antenna) for the statistics.

4.5.4 Calibration Smoothing (smoothcal)

The `smoothcal` task will smooth calibration solutions (most usefully G or T) over a longer time interval to reduce noise and outliers. The inputs are:

```
# smoothcal :: Smooth calibration solution(s) derived from one or more sources:
```

```
vis          =          '' # Name of input visibility file
```

```

tablein      =      '' # Input calibration table
caltable     =      '' # Output calibration table
field        =      '' # Field name list
smoothtype   =      'median' # Smoothing filter to use
smoothtime   =      60.0 # Smoothing time (sec)
async        =      False # if True run in the background, prompt is freed

```

The smoothing will use the `smoothtime` and `smoothtype` parameters to determine the new data points which will replace the previous points on the same time sampling grid as for the `tablein` solutions. The currently supported `smoothtype` options:

- `'mean'` — use the mean of the points within the window defined by `smoothtime` (a “boxcar” average),
- `'median'` — use the median of the points within the window defined by `smoothtime` (most useful when many points lie in the interval).

Note that `smoothtime` defines the width of the time window that is used for the smoothing.

ALERT: Note that `smoothcal` currently smooths by `field` and `spw`, and thus you cannot smooth solutions from different sources or bands together into one solution.

An example using the `smoothcal` task to smooth an existing table:

```

smoothcal('n4826_16apr.ms',
          tablein='n4826_16apr.gcal',
          caltable='n4826_16apr.smoothcal',
          smoothtime=7200.,
          smoothtype='mean')

# Plot up before and after tables
plotcal('n4826_16apr.gcal', '', 'amp', antenna='1', subplot=211)
plotcal('n4826_16apr.smoothcal', '', 'amp', antenna='1', subplot=212)

```

This example uses 2 hours (7200 sec) for the smoothing time and `smoothtype='mean'`. The `plotcal` results are shown in Figure 4.6.

4.5.5 Calibration Interpolation and Accumulation (`accum`)

The `accum` task is used to interpolate calibration solutions onto a different time grid, and to *accumulate* incremental calibrations into a *cumulative* calibration table.

Its inputs are:

```

# accum :: Accumulate incremental calibration solutions

vis          =      '' # Name of input visibility file
tablein      =      '' # Input (cumulative) calibration table; use '' on first run

```

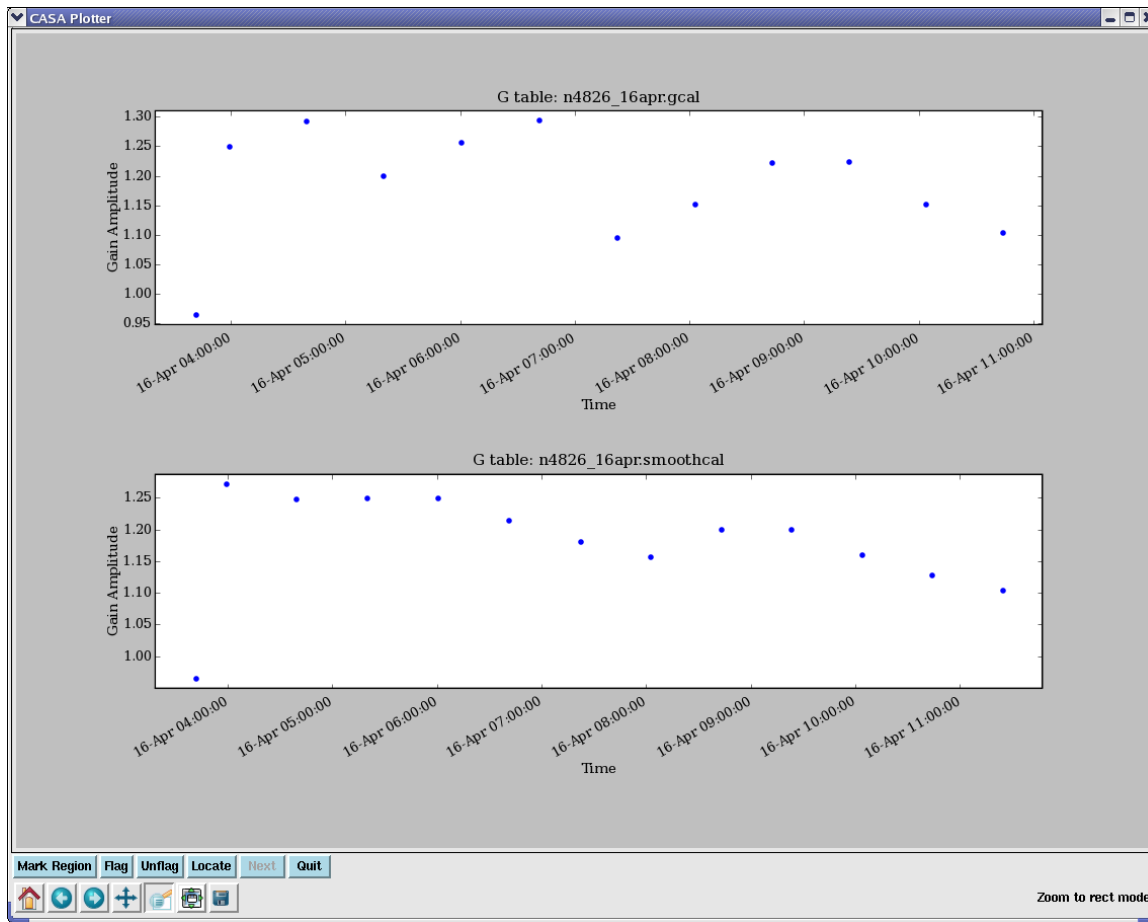



Figure 4.6: The 'amp' of gain solutions for NGC4826 before (top) and after (bottom) smoothing with a 7200 sec `smoothtime` and `smoothtype='mean'`. Note that the first solution is in a different `spw` and on a different source, and is not smoothed together with the subsequent solutions.

```

    accumtime =      1.0  # Timescale on which to create cumulative table

incrtable    =      ''   # Input incremental calibration table to add
caltable     =      ''   # Output (cumulative) calibration table
field        =      ''   # List of field names to process from tablein.
calfield     =      ''   # List of field names to use from incrtable.
interp       =      'linear' # Interpolation mode to use for resampling incrtable solutions
spwmap       =      [-1]  # Spectral window combinations to apply

```

The *mapping* implied here is

```
tablein + incrtable => caltable
```

(mathematically the cal solutions are multiplied as complex numbers as per the Measurement

Equation). The `tablein` is optional (see below). You must specify an `incrtable` and a `caltable`.

The `tablein` parameter is used to specify the existing cumulative calibration table to which an incremental table is to be applied. Initially, no such table exists, and if `tablein=''` then `accumulate` will generate one from scratch (on-the-fly), using the timescale (in seconds) specified by the sub-parameter `accumtime`. These nominal solutions will be unit-amplitude, zero-phase calibration, ready to be adjusted by accumulation according to the settings of other parameters. When `accumtime` is negative (the default), the table name specified in `tablein` must exist and will be used. If `tablein` is specified, then the entries in that table will be used.

The `incrtable` parameter is used to specify the incremental table that should be applied to `tablein`. The calibration type of `incrtable` sets the type assumed in the operation, so `tablein` (if specified) must be of the same type. If it is not, `accum` will exit with an error message. (Certain combinations of types and subtypes will be supported by `accum` in the future.)

The `caltable` parameter is used to specify the name of the output table to write. If un-specified (''), then `tablein` will be overwritten. Use this feature with care, since an error here will require building up the cumulative table from the most recent distinct version (if any).

The `field` parameter specifies those field names in `tablein` to which the incremental solution should be applied. The solutions for other fields will be passed to `caltable` unaltered. If the cumulative table was created from scratch in this run of `accumulate`, then the solutions for these other fields will be unit-amplitude, zero-phase, as described above.

The `calfield` parameter is used to specify the fields to select from `incrtable` to use when applying to `tablein`. Together, use of `field` and `calfield` permit completely flexible combinations of calibration accumulation with respect to fields. Multiple runs of `accum` can be used to generate a single table with many combinations. In future, a 'self' mode will be enabled that will simplify the accumulation of field-specific solutions.

The `spwmap` parameter gives the mapping of the spectral windows in the `incrtable` onto those in `tablein` and `caltable`. The syntax is described in § 4.4.1.4.

The `interp` parameter controls the method used for interpolation. The options are (currently): 'nearest', 'linear', and 'aipslin'. These are described in § 4.4.1.4. For most purposes, the 'linear' option should suffice.

We now describe the two uses of `accum`.

4.5.5.1 Interpolation using (`accum`)

Calibration solutions (most notably G or T) can be interpolated onto the timestamps of the science target observations using `accum`.

The following example uses `accum` to interpolate an existing table onto a new time grid:

```
accum(vis='n4826_16apr.ms',
      tablein='',
      accumtime=20.0,
```

```

incrtable='n4826_16apr.gcal',
caltable='n4826_16apr.20s.gcal',
interp='linear',
spwmap=[0,1,1,1,1,1])

plotcal('n4826_16apr.gcal','','phase',antenna='1',subplot=211)
plotcal('n4826_16apr.20s.gcal','','phase',antenna='1',subplot=212)

```

See Figure 4.7 for the `plotcal` results. The data used in this example is BIMA data (single polarization `YY`) where the calibrators were observed in single continuum spectral windows (`spw='0,1'`) and the target NGC4826 was observed in 64-channel line windows (`spw='2,3,4,5'`). Thus, it is necessary to use `spwmap=[0,1,1,1,1,1]` to map the bandpass calibrator in `spw='0'` onto itself, and the phase calibrator in `spw='1'` onto the target source in `spw='2,3,4,5'`.

4.5.5.2 Incremental Calibration using (`accum`)

It is occasionally desirable to solve for and apply calibration incrementally. This is the case when a calibration table of a certain type already exists (from a previous solve), a solution *of the same type* and incremental *relative to the first* is required, and it is not possible or convenient to recover the cumulative solution by a single solve.

Much of the time, it is, in fact, possible to recover the cumulative solution. This is because the equation describing the solution for the incremental solution (using the original solution), and that describing the solution for their product are fundamentally the same equation—the cumulative solution, if unique, must always be the same no matter what initial solution is. One circumstance where an incremental solution is necessary is the case of *phase-only* self-calibration relative to a full amplitude and phase calibration already obtained (from a different field).

For example, a phase-only '`G`' self-calibration on a target source may be desired to tweak the full amplitude and phase '`G`' calibration already obtained from a calibrator. The initial calibration (from the calibrator) contains amplitude information, and so must be carried forward, yet the phase-only solution itself cannot (by definition) recover this information, as a full amplitude and phase self-calibration would. In this case, the initial solution must be applied while solving for the phase-only solution, then the two solutions combined to form a cumulative calibration embodying the net effect of both. In terms of the Measurement Equation, the net calibration is the product of the initial and incremental solutions.

Cumulative calibration tables also provide a means of generating carefully interpolated calibration, on variable user-defined timescales, that can be examined prior to application to the data with `applycal`. The solutions for different fields and/or spectral windows can be interpolated in different ways, with all solutions stored in the same table.

The only difference between incremental and cumulative calibration tables is that incremental tables are generated directly from the calibration solving tasks (`gaincal`, `bandpass`, etc), and cumulative tables are generated from other cumulative and incremental tables via `accum`. In all

Other Packages:

The analog of `accum` in classic AIPS is the use of `CLCAL` to combine a series of (incremental) SN calibration tables to form successive (cumulative) CL calibration tables. AIPS SN/CL tables are the analog of '`G`' tables in CASA.

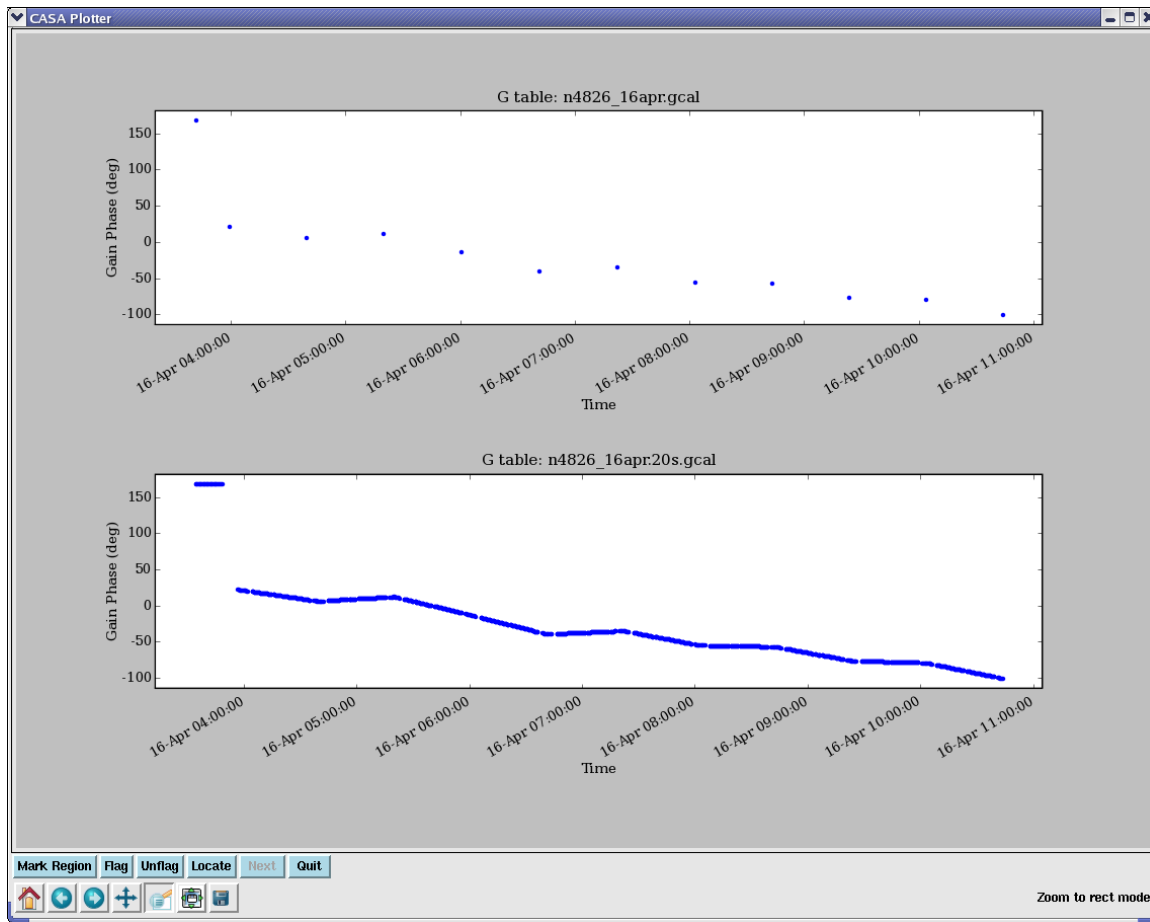


Figure 4.7: The 'phase' of gain solutions for NGC4826 before (top) and after (bottom) 'linear' interpolation onto a 20 sec `accumtime` grid. The first scan was 3C273 in `spw='0'` while the calibrator scans on 1331+305 were in `spw='1'`. The use of `spwmap` was necessary to transfer the interpolation correctly onto the NGC4826 scans.

other respects (internal format, application to data with `applycal`, plotting with `plotcal`, etc.), they are the same, and therefore interchangeable. Thus, `accumulate` and `cumulative` calibration tables need only be used when circumstances require it.

The `accum` task represents a generalization on the classic AIPS `CLCAL` (see sidebox) model of cumulative calibration in that its application is not limited to accumulation of 'G' solutions. In principle, any basic calibration type can be accumulated (onto itself), as long as the result of the accumulation (matrix product) is of the same type. This is true of all the basic types, except 'D'. Accumulation is currently supported for 'B', 'G', and 'T', and, in future, 'F' (ionospheric Faraday rotation), delay-rate, and perhaps others. Accumulation of certain specialized types (e.g.,

'GSPLINE', 'TOPAC', etc.) onto the basic types will be supported in the near future. The treatment of various calibration from ancillary data (e.g., system temperatures, weather data, WVR, etc.), as they become available, will also make use of accumulate to achieve the net calibration.

Note that accumulation only makes sense if treatment of a uniquely incremental solution is required (as described above), or if a careful interpolation or sampling of a solution is desired. In all other cases, re-solving for the type in question will suffice to form the net calibration of that type. For example, the product of an existing 'G' solution and an amplitude and phase 'G' self-cal (solved with the existing solution applied), is equivalent to full amplitude and phase 'G' self-cal (with no prior solution applied), as long as the timescale of this solution is at least as short as that of the existing solution.

One obvious application is to calibrate the amplitudes and phases on different timescales during self-calibration. Here is an example, using the Jupiter VLA 6m continuum imaging example (see Appendix F.2):

```
# Put clean model into MODEL_DATA column
ft(vis='jupiter6cm.usecase.split.ms',
   model='jupiter6cm.usecase.clean1.model')

# Phase only self-cal on 10s timescales
gaincal(vis='jupiter6cm.usecase.split.ms',
        caltable='jupiter6cm.usecase.phasecal1',
        gaintype='G',
        calmode='p',
        refant='6',
        solint=10.0,
        minsnr=1.0)

# Plot up solution phase and SNR
plotcal('jupiter6cm.usecase.phasecal1','','phase',antenna='1',subplot=211)
plotcal('jupiter6cm.usecase.phasecal1','','snr',antenna='1',subplot=212)

# Amplitude and phase self-cal on scans
gaincal(vis='jupiter6cm.usecase.split.ms',
        caltable='jupiter6cm.usecase.scancal1',
        gaintable='jupiter6cm.usecase.phasecal1',
        gaintype='G',
        calmode='ap',
        refant='6',
        solint='inf',
        minsnr=1.0)

# Plot up solution amp and SNR
plotcal('jupiter6cm.usecase.scancal1','','amp',antenna='1',subplot=211)
plotcal('jupiter6cm.usecase.scancal1','','snr',antenna='1',subplot=212)

# Now accumulate these - they will be on the 10s grid
accum(vis='jupiter6cm.usecase.split.ms',
      tablein='jupiter6cm.usecase.phasecal1',
```

```

incrtable='jupiter6cm.usecase.scancal1',
caltable='jupiter6cm.usecase.selfcal1',
interp='linear')

# Plot this up
plotcal('jupiter6cm.usecase.selfcal1','','amp',antenna='1',subplot=211)
plotcal('jupiter6cm.usecase.selfcal1','','phase',antenna='1',subplot=212)

```

The final plot is shown in Figure 4.8

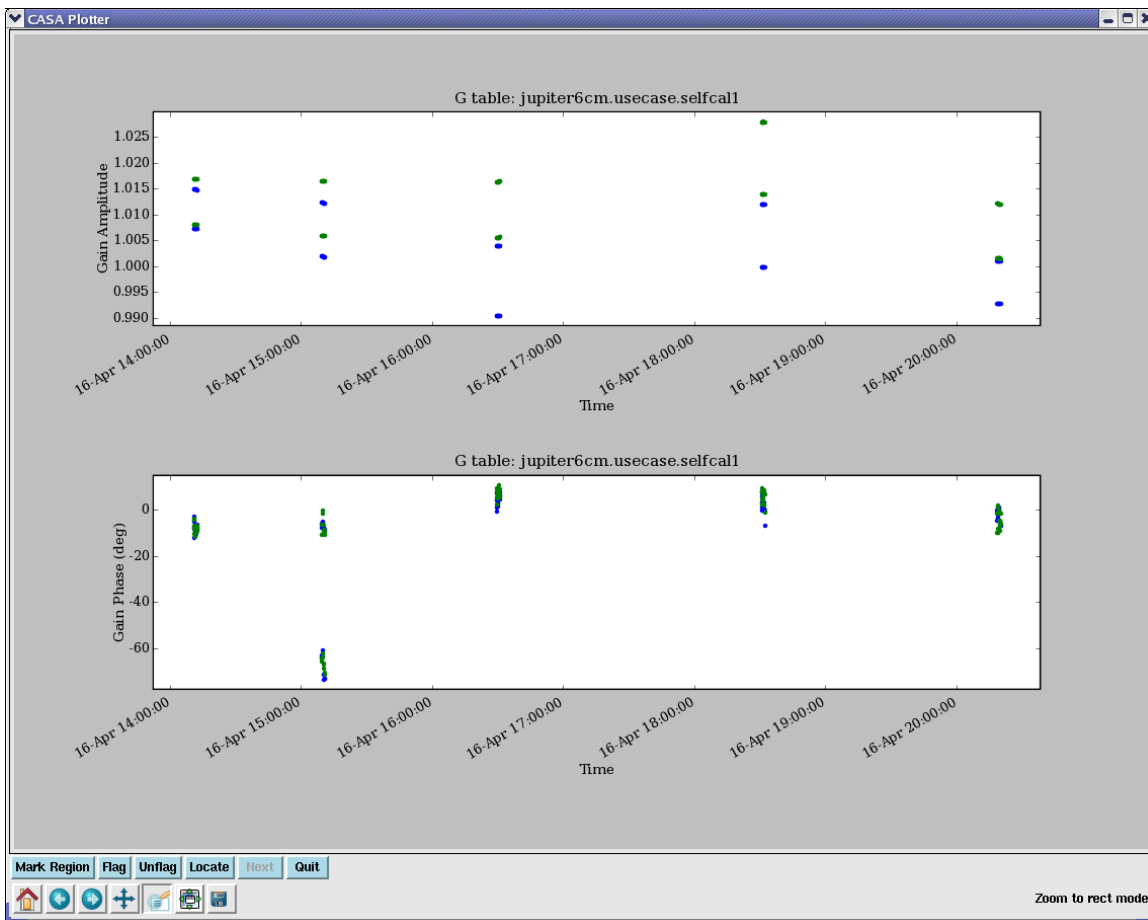


Figure 4.8: The final 'amp' (top) and 'phase' (bottom) of the self-calibration gain solutions for Jupiter. An initial phase calibration on 10s `solint` was followed by an incremental gain solution on each scan. These were accumulated into the cumulative solution shown here.

ALERT: Only interpolation is offered in `accum`, no smoothing (as in `smoothcal`).

4.6 Application of Calibration to the Data

After the calibration solutions are computed and written to one or more calibration tables, one then needs to apply them to the data.

4.6.1 Application of Calibration (applycal)

After all relevant calibration types have been determined, they must be applied to the target source(s) before splitting off to a new MS or before imaging. This is currently done by explicitly taking the data in the DATA column in the MAIN table of the MS, applying the relevant calibration tables, and creating the CORRECTED_DATA scratch column. The original DATA column is untouched.

The `applycal` task does this. The inputs are:

```
# applycal :: Apply calibration solution(s) to data

vis          =      '' # Name of input visibility file
field        =      '' # Names or indices of data fields to apply calibration ''==>all
spw          =      '' # spectral window:channels: ''==>all
selectdata   =      False # Other data selection parameters
gaintable    =      '' # List of calibration table(s) to apply
gainfield    =      '' # Field selection for each gaintable
interp       =      '' # Interpolation mode (in time) for each gaintable
spwmap       =      [] # Spectral window mapping for each gaintable (see help)
gaincurve    =      False # Apply VLA antenna gain curve correction
opacity      =      0.0 # Opacity correction to apply (nepers)
parang       =      False # Apply the parallactic angle correction
calwt        =      True  # Apply calibration also to the WEIGHTS
async        =      False # if True run in the background, prompt is freed
```

As in other tasks, setting `selectdata=True` will open up the other selection sub-parameters (see § 2.3). Many of the other parameters are the common calibration parameters that are described in § 4.4.1.

The single non-standard parameter is the `calwt` option to toggle the ability to scale the visibility weights by the inverse of the products of the scale factors applied to the amplitude of the antenna gains (for the pair of antennas of a given visibility). This should in *almost all cases* be set to its default (`True`). The weights should reflect the inverse noise variance of the visibility, and errors in amplitude are usually also in the weights.

Alert: *Current* (as of June 2010) EVLA data has no weights to the data. To avoid trouble, `calwt=False` should be set for those data sets. Older VLA data should still be calibrated with `calwt=True`.

For `applycal`, the list of final cumulative tables is given in `gaintable`. In this case you will have run `accum` if you have done incremental calibration for any of the types, such as 'G'. You can also feed `gaintable` the full sets and rely on use of `gainfield`, `interp` and `spwmap` to do the correct interpolation and transfer. It is often more convenient to go through accumulation of each type

with `accum` as described above (see § 4.5.5.2), as this makes it easier to keep track of the sequence of incremental calibration as it is solved and applied. You can also do any required smoothing of tables using `smoothcal` (§ 4.5.4), as this is not yet available in `accum` or `applycal`.

If you are not doing polarization calibration or imaging, then you can set `parang=False` to make the calculations faster. If you are applying polarization calibration, or wish to make polarization images, then set `parang=True` so that the parallactic angle rotation is applied to the appropriate correlations. Currently, you must do this in `applycal` as this cannot be done on-the-fly in `clean` or `mosaic`. See § 4.4.1.3 for more on `parang`.

For example, to apply the final bandpass and flux-scaled gain calibration tables solutions to the NGC5921 data:

```
default('applycal')

vis='ngc5921.usecase.ms'

# We want to correct the calibrators using themselves
# and transfer from 1445+099 to itself and the target N5921

# Start with the fluxscale/gain and bandpass tables
gaintable=['ngc5921.usecase.fluxscale','ngc5921.usecase.bcal']

# pick the 1445+099 (field 1) out of the gain table for transfer
# use all of the bandpass table
gainfield = ['1','*']

# interpolation using linear for gain, nearest for bandpass
interp = ['linear','nearest']

# only one spw, do not need mapping
spwmap = []

# all channels, no other selection
spw = ''
selectdata = False

# no prior calibration
gaincurve = False
opacity = 0.0

# select the fields for 1445+099 and N5921 (fields 1 and 2)
field = '1,2'

applycal()

# Now for completeness apply 1331+305 (field 0) to itself

field = '0'
gainfield = ['0','*']
```



```
applycal()
```

```
# The CORRECTED_DATA column now contains the calibrated visibilities
```

In another example, we apply the final cumulative self-calibration of the Jupiter continuum data obtained in the example of § 4.5.5.2:

```
applycal(vis='jupiter6cm.usecase.split.ms',
         gaintable='jupiter6cm.usecase.selfcal1',
         selectdata=False)
```

Again, it is important to remember the relative nature of each calibration term. A term solved for in the presence of others is, in effect, residual to the others, and so must be used in combination with them (or new versions of them) in subsequent processing. At the same time, it is important to avoid isolating the same calibration effects in more than one term, e.g., by solving for both 'G' and 'T' separately (without applying the other), and then using them together.

It is always a good idea to examine the corrected data after calibration (using `plotxy` to compare the raw ('data') and corrected ('corrected') visibilities), as we describe next.

4.6.2 Examine the Calibrated Data

Once the source data is calibrated using `applycal`, you should examine the *uv* data and flag anything that looks bad. If you find source data that has not been flanked by calibration scans, delete it (it will not be calibrated).

For example, to look at the calibrated Jupiter data in the last example given in the previous section:

```
plotxy('jupiter6cm.usecase.split.ms', 'uvdist', 'amp', 'corrected',
       selectdata=True, correlation='RR LL', fontsize = 14.0)
```

will show the `CORRECTED_DATA` column. See Figure 4.9.

See § 3.4 for a description of how to display and edit data using `plotms` or `plotxy`, and § 7.4 for use of the viewer to visualize and edit a Measurement Set.

4.6.3 Resetting the Applied Calibration using (`clearcal`)

The `applycal` task will set the `CORRECTED_DATA` column. The `clearcal` task will reset it to be the same as the `DATA` column. This may or may not be what you really want to do — nominally you will rerun `applycal` to get new calibration if you have changed the tables or want to apply them differently.

There is only a single input to `clearcal`:

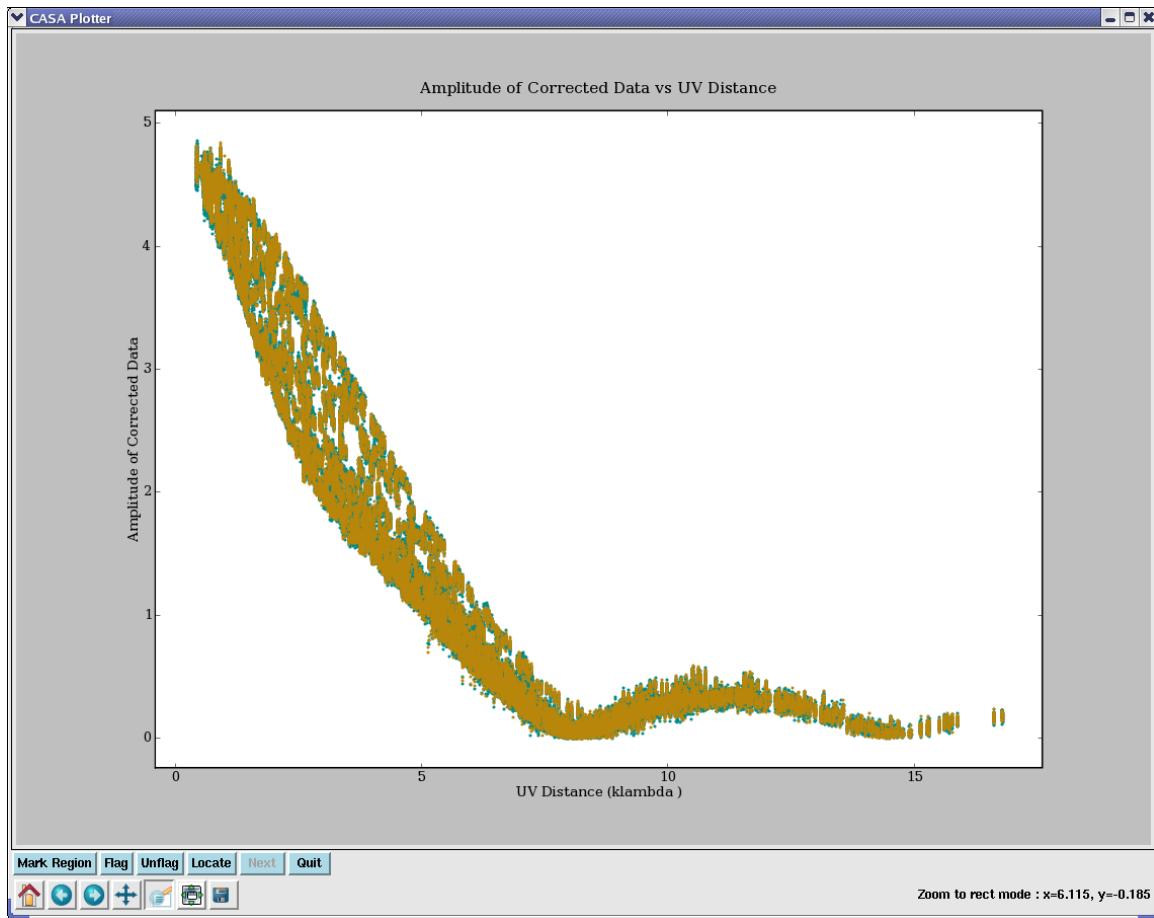


Figure 4.9: The final 'amp' versus 'uvdist' plot of the self-calibrated Jupiter data, as shown in plotxy. The 'RR LL' correlations are selected. No outliers that need flagging are seen.

```
# clearcal :: Re-initializes calibration for an ms
vis          =          ''          # Name of input visibility file
```

Note: clearcal also resets the MODEL_DATA column to (1,0) for all fields and spectral windows.

4.7 Other Calibration and UV-Plane Analysis Options

4.7.1 Splitting out Calibrated uv data (split)

The `split` task will apply calibration and output a new sub-MS containing a specified list of sources (usually a single source). The inputs are:

```

# split :: Create a visibility subset from an existing visibility set:
vis      =      '' # Name of input measurement set
outputvis =      '' # Name of output measurement set
datacolumn = 'corrected' # Which data column(s) to split out
field    =      '' # Select field using field id(s) or field name(s)
spw      =      '' # Select spectral window/channels
width    =      1 # Number of channels to average to form one output channel
antenna  =      '' # Select data based on antenna/baseline
timebin  =      '0s' # Value for timeaveraging
timerange =      '' # Select data based on time range
scan     =      '' # Select data based on scan numbers
array    =      '' # Select (sub)array by array ID number(s)
uvrange  =      '' # Select data based on uv distance range
async    =      False # If true the taskname must be started using split(...)

```

Usually you will run `split` with `datacolumn='corrected'` as previous operations (e.g. `applycal`) will have placed the calibrated data in the `CORRECTED_DATA` column of the MS. This will produce a new MS with this corrected data in its `DATA` column. The modes available in `datacolumn` are:

```

'data', 'model', 'corrected',           # produce MS with single DATA column
'data,model', 'data,corrected', 'model,corrected', # pairs of columns
'all'                                     # all columns 'data,model,corrected'

```

We recommend sticking to the simple single-column modes (e.g. `'data'` or `'corrected'`) or `'all'` if all columns are in the MS. Further processing may get confused by mismatched pairs of columns.

For example, to split out 46 channels (5-50) from `spw 1` of our NGC5921 calibrated dataset:

```

split(vis='ngc5921.usecase.ms',
      outputvis='ngc5921.split.ms',
      field='2',           # Output NGC5921 data (field 2)
      spw='0:5~50',       # Select 46 chans from spw 0
      datacolumn='corrected') # Take the calibrated data column

```

4.7.1.1 Averaging in split

Time and channel averaging are now available using the `timebin` and `width` parameters.

The `timebin` parameter give the averaging time. It takes a quantity, e.g.

```
timebin = '30s'
```

The `width` parameter defines the number of channels to average to form a given output channel. This can be specified globally for all `spw`, e.g.

```
width = 5
```

or specified per `spw`, e.g.

```
width = [2,3]
```

to average 2 channels of 1st spectral window selected and 3 in the second one.

ALERT: The ability to average channels in both time and channel simultaneously is not yet available. Also, if you average time and channel through sequential runs of `split`, you must average in time first.

4.7.2 Hanning smoothing of uv data (`hanningsmooth`)

The `hanningsmooth` task will apply Hanning smoothing to a spectral line uv data set. It will be applied to the data in the `DATA` column of the input MS and it writes the Hanning smoothed data into the `CORRECTED_DATA` column of that same MS.

Hanning smoothing replaces the contents of channel i with a weighted sum of the contents of a number of channels surrounding channel i . In its current form, only channels $i-1$, i , and $i+1$ participate, with weights 0.25, 0.50, and 0.25 respectively, but we intend to extend the kernel size in future releases. A typical use for Hanning smoothing is to remove Gibbs ringing.

The inputs are:

```
# hanningsmooth :: Hanning smooth frequency channel data
vis                = 'ngc5921.split.ms'      # Name of input visibility file (MS)
async              =          False
```

In many cases the data to be smoothed are in the `CORRECTED_DATA` column of the MS; in that case, run `split` first to copy the contents of the `CORRECTED_DATA` column of the input MS to the `DATA` column of the output MS. Then run `hanningsmooth` on the newly created MS.

After hanning smoothing, the contents of the first and last channel of each visibility are undefined; `hanningsmooth` will therefore flag the first and last channel.

ALERT: We intend to make the kernel size a user supplied parameter. In the longer term we intend to offer other varieties of spectral smoothing as well.

4.7.3 Model subtraction from uv data (`uvsb`)

The `uvsb` task will subtract the value in the `MODEL` column from that in the `CORRECTED_DATA` column in the input MS and store the result in that same `CORRECTED_DATA` column.

The reverse operation is achieved by specifying `reverse = True`: in that case `uvsb` will add the value in the `MODEL` column to that in the `CORRECTED_DATA` column in the input MS and store the result in that same `CORRECTED_DATA` column.

The inputs are:

```
# uvsub :: Subtract/add model from/to the corrected visibility data.

vis      =      '' # Name of input visibility file (MS)
reverse  =      False # reverse the operation (add rather than subtract)
async    =      False
```

For example:

```
uvsub('ngc5921.split.ms')
```

ALERT: Currently, `uvsub` operates on the scratch columns in the MS `vis`. Eventually we will provide the option to handle these columns behind the scenes and to write out a new MS.

4.7.4 UV-Plane Continuum Subtraction (`uvcontsub`)

At this point, consider whether you are likely to need continuum subtraction. If there is significant continuum emission present in what is intended as a spectral line observation, continuum subtraction may be desirable. You can estimate and subtract continuum emission in the uv -plane prior to imaging or wait and subtract an estimate of it in the image-plane. Note that neither method is ideal, and the choice depends primarily upon the distribution and strength of the continuum emission. Subtraction in the uv -plane is desirable if continuum emission dominates the source, since deconvolution of the line emission will be more robust if not subject to errors in deconvolution of the brighter continuum. There is also a performance benefit since the continuum is probably the same in each channel of the observation, and it is desirable to avoid duplication of effort. However, the main drawback of subtraction in the uv -plane is that it is only strictly correct for the phase center, since without the Fourier transform, the visibilities only describe the phase center. Thus, uv -plane continuum subtraction will be increasingly poor for emission distributed further from the phase center. If the continuum emission is relatively weak, it is usually adequate to subtract it in the image plane; this is described in the Image Analysis section of this cookbook. Here, we describe how to do continuum subtraction in the uv -plane.

The uv -plane continuum subtraction is performed by the `uvcontsub` task. First, determine which channels in your data cube do not have line emission, perhaps by forming a preliminary image as described in the next chapter. This image will also help you decide whether or not you need to come back and do uv -plane continuum subtraction at all.

ALERT: Even when setting `splitdata=True` this task will overwrite what is in the `CORRECTED_DATA` column of the MS. We strongly urge you to make a copy of the MS using `split` (e.g. with `datacolumn='corrected'`) and then work on that in order to keep from altering the original dataset.

The inputs to `uvcontsub` are:

```
# uvcontsub :: Continuum fitting and subtraction in the uv plane
vis      =      '' # Nome of input visibility file
field    =      '' # Select field using field id(s) or field name(s)
```

```

fitspw    =      ''    # Spectral window/channel selection for fitting the continuum
spw       =      ''    # Spectral window selection for subtraction/export
solint    =      'int'  # Continuum fit timescale
fitorder  =      0     # Polynomial order for the fit
fitmode   =      'subtract' # Use of continuum fit (subtract,replace,model)
splitdata =      False  # Split out continuum, continuum-subtracted data
async    =      False

```

For each baseline, and over the timescale specified in `solint`, `uvcontsub` will provide a simple linear fit to the real and imaginary parts of the (continuum-only) channels specified in `fitspw` (using the standard `spw` selection syntax), and then subtract this model from all channels specified in `spw`, or from all channels in spectral windows of `fitspw` if `spw=''`. **ALERT:** The fits are currently done independently in the spectral windows specified in `fitspw`, and thus overlapping channels in different `spw` will be corrected only with the fits from their respective `fitspw`.

Usually, one would set `solint='int'` which does no averaging and fits each integration. However, if the continuum emission comes from a small region around the phase center, then you can set `solint` larger (as long as it is shorter than the timescale for changes in the visibility function of the continuum). If your scans are short enough you can also use scan averaging `solint='inf'`. Be warned, setting `solint` too large will introduce “time smearing” in the estimated continuum and thus not properly subtracting emission not at the phase center.

Running `uvcontsub` with `fitmode='subtract'` will replace the `CORRECTED_DATA` column in the MS with continuum-subtracted line data and the `MODEL_DATA` column with the continuum model. You can use `fitmode='replace'` to replace the `CORRECTED_DATA` column with the continuum model; however, it is probably better to use `fitmode='subtract'` and then use `split` to select the `MODEL_DATA` and form a dataset appropriate for forming an image of the estimated continuum. Note that a continuum image formed from this model will only be strictly correct near the phase center, for the reasons described above.

The `splitdata` parameter can be used to have `uvcontsub` write out split MS for both the continuum-subtracted data and the continuum. It will leave the input MS in the state as if `fitmode='subtract'` was used. Note that the entire channel range of the MS will be written out (not just the channels specified in `spw` that have had the subtraction), so follow up with a `split` if you want to further restrict the output channel range. If `splitdata=True`, then `uvcontsub` will make two output MS with names `<input msname>.contsub` and `<input msname>.cont`. **ALERT:** be sure to run with `fitmode='subtract'` if setting `splitdata=True`.

As mentioned above, it is currently the case that `uvcontsub` will overwrite the `CORRECTED_DATA` column. Therefore, it is desirable to first `split` the relevant corrected data into a new Measurement Set. If you run `uvcontsub` on the original dataset, you will have to re-apply the calibration as described in the previous chapter.

So, the recommended procedure is as follows:

- Finish calibration as described in the previous chapter.
- Use `split` to form a separate dataset.

- Use the `invert` or `clean` task on the `split` result to form an exploratory image that is useful for determining the line-free channels.
- Use `uvcontsub` with `mode='subtract'` to subtract the continuum from the `CORRECTED_DATA` in the MS, and write the continuum model in the `MODEL_DATA` column. Set `splitdata=True` to have it automatically split out continuum-subtracted and continuum datasets, else do this manually.
- Image the line-only emission with the `clean` task.
- If an image of the estimated continuum is desired, and you did not use `splitdata=True`, then run `split` again (on the `uvcontsub`'d dataset), and select the `MODEL_DATA`; then run `clean` to image it.

For example, we perform uv-plane continuum subtraction on our NGC5921 dataset:

```
# Want to use channels 4-6 and 50-59 for continuum
uvcontsub(vis='ngc5921.usecase.ms',
          field='N5921*',
          spw='', # all spw (only 0 in this data)
          fitspw='0:4~7;50~59' # channels 4-6 and 50-59
          solint='inf', # scans are short enough
          fitorder=0 # mean only
          fitmode='subtract' # uv-plane subtraction
          splitdata=True) # split the data for us

# You will see it made two new MS:
# ngc5921.usecase.ms.cont
# ngc5921.usecase.ms.contsub
```

4.7.5 Spectral regridding of the MS (`cvel`)

Although not strictly a calibration operation, spectral regridding of a MS is available to aid in calibration operations (e.g. continuum subtraction) and preparation for imaging. For this purpose, the `cvel` task has been developed.

The inputs are:

```
# cvel :: regrid an MS to a new spectral window / channel structure or frame
vis          = '' # Name of input measurement set
outputvis    = '' # Name of output measurement set
passall      = False # Pass through (to output MS) non-selected data with no change
field        = '' # Select field using field id(s) or field name(s)
spw          = '' # Select spectral window/channels
selectdata   = False # Other data selection parameters
mode         = 'channel' # Regridding mode
  nchan      = -1 # Number of channels in output spw (-1=all)
  start      = 0 # first input channel to use
  width      = 1 # Number of input channels to average
```

```

interpolation = 'linear' # Spectral interpolation method

phasecenter   = ''      # Image phase center: position or field index
restfreq      = ''      # rest frequency (see help)
outframe      = ''      # Output frame (''=keep input frame)
veltype       = 'radio' # velocity definition
hanning       = False   # Turn on Hanning smoothing of spectral channels
async        = False   # If true the taskname must be started using cvel(...)

```

The key parameters for the operation of `cvel` are the regridding mode, the output reference `outframe` and `veltype`, and the standard selection parameters (in particular `spw` and `field`).

The syntax for mode options ('channel', 'velocity', 'frequency', 'channel.b') has been made compatible with the respective modes of `clean` (§ 5.2.5). The combination of selected `spw` and `mode` will determine the output channels and `spw(s)`:

```

spw = '0,1'; mode = 'channel'
# will produce a single spw containing all channels in spw 0 and 1
spw='0:5~28^2'; mode = 'channel'
# will produce a single spw made with channels (5,7,9,...,25,27)
spw = '0'; mode = 'channel': nchan=3; start=5; width=4
# will produce an spw with 3 output channels
# new channel 1 contains data from channels (5+6+7+8)
# new channel 2 contains data from channels (9+10+11+12)
# new channel 3 contains data from channels (13+14+15+16)
spw = '0:0~63^3'; mode='channel'; nchan=21; start = 0; width = 1
# will produce an spw with 21 channels
# new channel 1 contains data from channel 0
# new channel 2 contains data from channel 2
# new channel 21 contains data from channel 61
spw = '0:0~40^2'; mode = 'channel'; nchan = 3; start = 5; width = 4
# will produce an spw with three output channels
# new channel 1 contains channels (5,7)
# new channel 2 contains channels (13,15)
# new channel 3 contains channels (21,23)

```

The simplest use of `cvel` is to shift a single spectral window into an output frame without regridding. This is done with `mode='channel'`. For example:

```

cvel(vis='test_w3oh_nohann.ms',
     outputvis='test_w3oh_nohann_chanbary.ms',
     mode='channel', nchan=-1, start=0, width=1,
     interpolation='nearest',
     phasecenter='',
     spw='',
     restfreq='1665.4018MHz',
     outframe='BARY')

```

does this for an observation of the OH line.

There is also a special `mode='channel_b'` that does not force a linear output frequency grid, e.g. for irregularly spaced/overlapping spectral windows), but is nominally faster. This is not equivalent to a `clean` output gridding mode, although `clean` will work on this spectral lattice.

We recommend using the `mode='velocity'` and `mode='frequency'` options, as it is easiest to determine what the resulting channelization will be. For example:

```
cvel(vis='test_w3oh_nohann.ms',
     outputvis='test_w3oh_nohann_cvellsrk.ms',
     mode='velocity',nchan=45,start='-35.0km/s',width='-0.55km/s',
     interpolation='linear',
     phasecenter='',
     spw='',
     restfreq='1665.4018MHz',
     outframe='LSRK')

cvel(vis='test_w3oh_nohann.ms',
     outputvis='test_w3oh_nohann_cvelbary.ms',
     mode='velocity',nchan=45,start='-35.0km/s',width='-0.55km/s',
     interpolation='linear',
     phasecenter='',
     spw='',
     restfreq='1665.4018MHz',
     outframe='BARY')
```

will transform a MS into the LSRK and BARYcenter frames respectively.

4.7.6 UV-Plane Model Fitting (`uvmodelfit`)

It is often desirable to fit simple analytic source component models directly to visibility data. Such fitting has its origins in early interferometry, especially VLBI, where arrays consisted of only a few antennas and the calibration and deconvolution problems were poorly constrained. These methods overcame the calibration uncertainties by fitting the models to calibration-independent closure quantities and the deconvolution problem by drastically limiting the number of free parameters required to describe the visibilities. Today, even with larger and better calibrated arrays, it is still desirable to use visibility model fitting in order to extract geometric properties such as the positions and sizes of discrete components in radio sources. Fits for physically meaningful component shapes such as disks, rings, and optically thin spheres, though idealized, enable connecting source geometry directly to the physics of the emission regions.

Visibility model fitting is carried out by the `uvmodelfit` task. The inputs are:

```
# uvmodelfit :: Fit a single component source model to the uv data:

vis      =      '' # Name of input visibility file
field    =      '' # field name or index
spw      =      '' # spectral window
```

```

selectdata = False # Activate data selection details
niter      = 5     # Number of fitting iterations to execute
comptype   = 'P'   # Component type (P=pt source,G=ell. gauss,D=ell. disk)
sourcepar  = [1, 0, 0] # Starting guess (flux,xoff,yoff,bmajaxrat,bpa)
varypar    = []    # Which parameters can vary in fit
outfile    = ''    # Optional output component list table
async      = False # if True run in the background, prompt is freed

```

ALERT: This task currently only fits a single component.

The user specifies the number of non-linear solution iterations (`niter`), the component type (`comptype`), an initial guess for the component parameters (`sourcepar`), and optionally, a vector of Booleans selecting which component parameters should be allowed to vary (`fixpar`), and a filename in which to store a CASA componentlist for use in other applications (`file`). Allowed `comptypes` are currently point 'P' or Gaussian 'G'.

The function returns a vector containing the resulting parameter list. This vector can be edited at the command line, and specified as input (`sourcepar`) for another round of fitting.

The `sourcepar` parameter is currently the only way to specify the starting parameters for the fit. For points, there are three parameters: I (total flux density), and relative direction (RA, Dec) offsets (in arcsec) from the observation's phase center. For Gaussians, there are three additional parameters: the Gaussian's semi-major axis width (arcsec), the aspect ratio, and position angle (degrees). It should be understood that the quality of the result is very sensitive to the starting parameters provided by the user. If this first guess is not sufficiently close to the global χ^2 minimum, the algorithm will happily converge to an incorrect local minimum. In fact, the χ^2 surface, as a function of the component's relative direction parameters, has a shape very much like the inverse of the absolute value of the dirty image of the field. Any peak in this image (positive or negative) corresponds to a local χ^2 minimum that could conceivably capture the fit. It is the user's responsibility to ensure that the correct minimum does the capturing.

Currently, `uvmodelfit` relies on the likelihood that the source is very near the phase center (within a beamwidth) and/or the user's savvy in specifying the starting parameters. This fairly serious constraint will soon be relieved somewhat by enabling a rudimentary form of uv-plane weighting to increase the likelihood that the starting guess is on a slope in the correct χ^2 valley.

Improvements in the works for visibility model fitting include:

- User-specifiable uv-plane weighting
- Additional component shapes, including elliptical disks, rings, and optically thin spheroids.
- Optional calibration pre-application
- Multiple components. The handling of more than one component depends mostly on efficient means of managing the list itself (not easy in command line options), which are currently under development.
- Combined component and calibration fitting.

Example (see Figure 4.10):

```
#
# Note: It's best to channel average the data if many channels
# before running a modelfit
#
split('ngc5921.ms','1445+099_avg.ms',
      datacolumn='corrected',field='1445*',width='63')

# Initial guess is that it's close to the phase center
# and has a flux of 2.0 (a priori we know it's 2.47)
uvmodelfit('1445+099_avg.ms',      # use averaged data
           niter=5,                # Do 5 iterations
           comptype='P',           # P=Point source, G=Gaussian, D=Disk
           sourcepar=[2.0,.1,.1], # Source parameters for a point source
           spw='0',               #
           outfile='gcal.cl')     # Output component list file

# Output looks like:
There are 19656 - 3 = 19653 degrees of freedom.
iter=0:  reduced chi2=0.0418509:  I=2,  dir=[0.1, 0.1] arcsec
iter=1:  reduced chi2=0.003382:   I=2.48562,  dir=[-0.020069, -0.0268826] arcsec
iter=2:  reduced chi2=0.00338012: I=2.48614,  dir=[0.00323428, -0.00232235] arcsec
iter=3:  reduced chi2=0.00338012: I=2.48614,  dir=[0.00325324, -0.00228963] arcsec
iter=4:  reduced chi2=0.00338012: I=2.48614,  dir=[0.00325324, -0.00228963] arcsec
iter=5:  reduced chi2=0.00338012: I=2.48614,  dir=[0.00325324, -0.00228963] arcsec
If data weights are arbitrarily scaled, the following formal errors
will be underestimated by at least a factor sqrt(reduced chi2). If
the fit is systematically poor, the errors are much worse.
I = 2.48614 +/- 0.0176859
x = 0.00325324 +/- 0.163019 arcsec
y = -0.00228963 +/- 0.174458 arcsec
Writing componentlist to file: /home/sandrok/smyers/Testing/Patch2/N5921/gcal.cl

# Fourier transform the component list into MODEL_DATA column of the MS
ft('1445+099_avg.ms', complist='gcal.cl')

# Plot data versus uv distance
plotxy('1445+099_avg.ms', xaxis='uvdist', datacolumn='corrected')

# Specify green circles for model data (overplotted)
plotxy('1445+099_avg.ms', xaxis='uvdist', datacolumn='model',
       overplot=True, plotsymbol='go')
```

4.8 Examples of Calibration

See the scripts provided in Appendix F for examples of calibration. In particular, we refer the interested user to the demonstrations for:

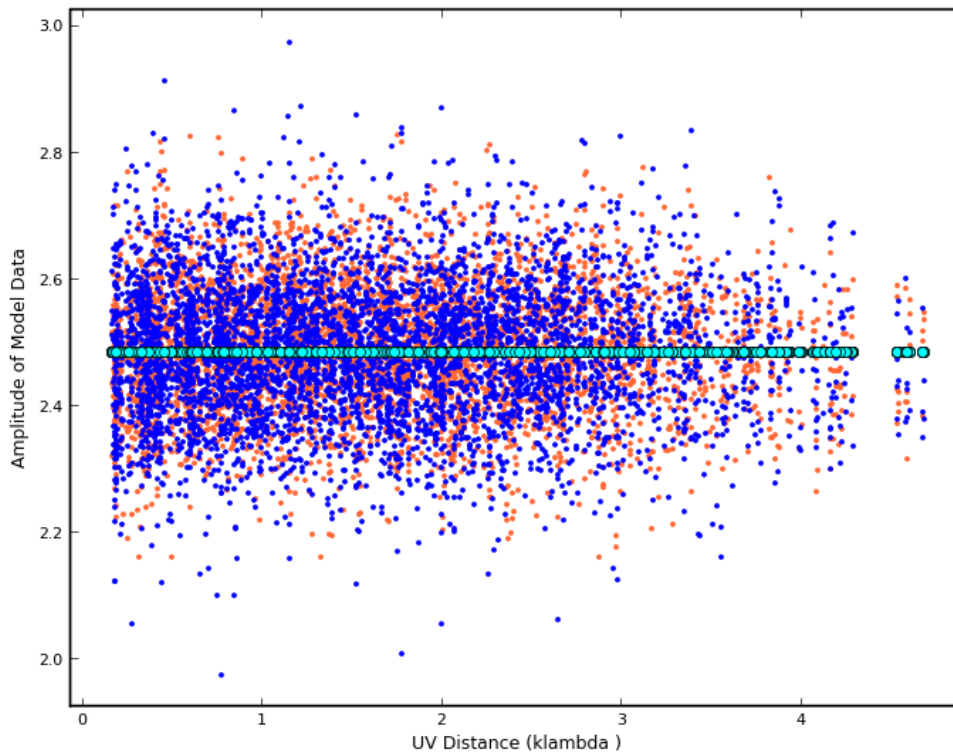


Figure 4.10: Use of plotxy to display corrected data (red and blue points) and uv model fit data (green circles).

- NGC5921 (VLA HI) — a quick demo of basic CASA spectral line calibration (F.1)
- Jupiter (VLA 6cm continuum polarimetry) — polarization calibration (F.2)

Chapter 5

Synthesis Imaging

This chapter describes how to make and deconvolve images starting from calibrated interferometric data, possibly supplemented with single-dish data or an image made from single-dish data. This data must be available in CASA (see § 2 on importing data). See § 4 for information on calibrating synthesis data. In the following sections, the user will learn how to make various types of images from synthesis data, reconstruct images of the sky using the available deconvolution techniques, include single-dish information in the imaging process, and to prepare to use the results of imaging for improvement of the calibration process (“self-calibration”).

Inside the Toolkit:

The `im` tool handles synthesis imaging operations.

5.1 Imaging Tasks Overview

The current imaging and deconvolution tasks are:

- `clean` — calculate a deconvolved image with a selected clean algorithm, including mosaicing, or make a dirty image (§ 5.3),
- `feather` — combine a single dish and synthesis image in the Fourier plane (§ 5.4),
- `deconvolve` — image-plane only deconvolution based on the dirty image and beam, using one of several algorithms (§ 5.7),
- `autoclean` — an experimental prototype task to automatically clean a dataset (§ 5.9).

There are also tasks that help you set up the imaging or interface imaging with calibration:

- `boxit` - create “cleanbox” deconvolution regions automatically from an image (§ 5.5.1),
- `makemask` - create “cleanbox” deconvolution regions from a box specification or file (§ 5.5.2),

- `ft` - Fourier transform the specified model (or component list) and insert the source model into the MODEL column of a visibility set (§ 5.6).

The full “tool kit” that allows expert-level imaging must still be used if you do not find enough functionality within the tasks above.

Information on other useful tasks and parameter setting can be found in:

- `listobs` — list whats in a MS (§ 2.2.5),
- `split`— Write out new MS containing calibrated data from a subset of the original MS (§ section:cal.split),
- `cvel` — regrid a spectral MS onto a new frequency channel system (§ 4.7.5),
- data selection — general data selection syntax (§ 2.3),
- `viewer` — image display including region statistics and image cube slice and profile capabilities (§ 7).

5.2 Common Imaging Task Parameters

We now describe some parameters are are common to the imaging tasks. These should behave the same way in any imaging task that they are found in. These are in alphabetical order.

ALERT: There are still a subset of data selection parameters used in the imaging tasks: `field`, `spw`, `timerange`. In a later patch, we will use the standard data selection set (§ 2.3).

Inside the Toolkit:
The `im.setimage` method is used to set many of the common image parameters. The `im.advise` method gives helpful advice for setting up for imaging.

5.2.1 Parameter `cell`

The `cell` parameter defines the pixel size in the x and y axes for the output image. If given as floats or integers, this is the cell size in arc seconds, e.g.

```
cell=[0.5,0.5]
```

make 0.5'' pixels. You can also give the cell size in *quantities*, e.g.

```
cell=['1arcmin', '1arcmin']
```

If a single value is given, then square pixels of that size are assumed.

5.2.2 Parameter field

The `field` parameter selects the field indexes or names to be used in imaging. Unless you are making a mosaic, this is usually a single index or name:

```
field = '0'           # First field (index 0)
field = '1331+305'   # 3c286
field = '*'          # all fields in dataset
```

The syntax for `field` selection is given in § 2.3.2.

5.2.3 Parameter imagename

The value of the `imagename` parameter is used as the root name of the output image. Depending on the particular task and the options chosen, one or more images with names built from that root will be created. For example, the `clean` task run with `imagename='ngc5921'` a series of output images will be created with the names `ngc5921.clean`, `ngc5921.residual`, `ngc5921.model`, etc.

If an image with that name already exists, it will in general be overwritten. Beware using names of existing images however. If the `clean` is run using an `imagename` where `<imagename>.residual` and `<imagename>.model` already exist then `clean` will continue starting from these (effectively restarting from the end of the previous `clean`). Thus, if multiple runs of `clean` are run consecutively with the same `imagename`, then the cleaning is incremental (as in the `difmap` package).

5.2.4 Parameter imsize

The image size in numbers of pixels on the x and y axes is set by `imsize`. For example,

```
imsize = [288, 288]
```

makes a square image 288 pixels on a side. If a single value is given, then a square image of that dimension is made. This need not be a power of two, but for optimal performance should be a “composite” number divisible only by 2 and also 3 and/or 5. Note that in the example above $288 = 2^5 \cdot 3^2$.

ALERT: You will be warned if you give a non-composite `imsize` and it will suggest the nearest appropriate value. But it will continue cleaning so you may have to abort if you want to make use of this suggestion. This restriction will be lifted in the future when we migrate to a better FFTW engine.

5.2.5 Parameter mode

The `mode` parameter defines how the frequency channels in the synthesis MS are mapped onto the image. The allowed values are: `mfs`, `channel`, `velocity`, `frequency`. The `mode` parameter is expandable, with some options uncovering a number of sub-parameters, depending upon its value.

5.2.5.1 Mode mfs

```

mode           =      'mfs'   # Spectral image definition (mfs,channel,velocity,frequency)
  nterms       =          1   # Number of Taylor terms for modeling the sky frequency dependence
  reffreq      =          ''   # Reference frequency for MFS (relevant only if nterms > 1)

```

The default `mode='mfs'` emulates multi-frequency synthesis in that each visibility-channel datum k with baseline vector \mathbf{B}_k at wavelength λ_k is gridded into the uv-plane at $\mathbf{u}_k = \mathbf{B}_k/\lambda_k$. The result is one or more images (depending on `nterms`), regardless of how many channels are in the input dataset. The first image plane is at the frequency given by the midpoint between the highest and lowest frequency channels in the input `spw(s)`. Currently, there is no way to choose the center frequency of the output image plane independently.

Higher order MFS (using `nterms>1`) is now available in CASA imaging. This does a Taylor expansion in frequency suitable for wide-band narrow-field imaging. The default `nterms=1` is equivalent to assuming no frequency dependence, while `nterms=2` is equivalent to the Sault-Wieringa algorithm¹ The `reffreq` parameter sets the reference frequency about which the Taylor expansion is done.

5.2.5.2 Mode channel

ALERT: Note that `mode='channel'` is intended as a shortcut to produce a cube based on the input MS channelization. It will be in the frame of the input MS. We recommend that users instead use the `'velocity'` and `'frequency'` modes which will produce cubes in other frames with more control of the cube spacing. These modes have defaults that will work from the MS spacing, reproducing the action of `mode='channel'`.

If `mode='channel'` is chosen, then an image cube will be created. This is an expandable parameter, with dependent parameters:

```

mode           =      'channel' # Spectral image definition(mfs, channel, velocity,frequency)
  nchan        =         -1   # Number of channels (planes) in output image
  start        =          0   # first input channel to use
  width        =          1   # Number of input channels to average
  interpolation =      'linear' # Spectral interpolation(nearest, linear, cubic)

```

The default `nchan=-1` will automatically produce a cube with the number of channels needed to span the (regridded) spectral windows of the MS. **ALERT:** This often results in extra blank channels at the beginning and end of the image cube, so it is usually more precise to specify `nchan` and `start` to get what you want.

The channelization of the resulting image is determined by the channelization in the MS of `vis` of the first `spw` specified (the “reference `spw`”). The actual channels to be gridded and used in the clean are selected via the `spw` parameter as usual. The resulting image cube will have `nchan` channels spaced evenly in frequency. The first output channel will be located at the frequency of

¹R.J. Sault & M.H. Wieringa, *Astron. Astrophys. Suppl.*, 108, 585–594 (1994).

channel `start` in the (first) reference `spw` (independent of what channels are selected using `spw`). If `width > 1`, then input MS channels with centers within a frequency range given by $(\text{width} + 1)/2$ times the reference `spw` spacing will be gridded together (as in `mode = 'mfs'` above) into the channels of the output image cube. The output channel spacing is thus given by `width` channels in the reference `spw` of the MS.

The `interpolation` sub-parameter (§ 5.2.5.5) sets how channels are gridded into the image cube planes. For `'nearest'`, the channels in `spw` beyond the first are mapped into the nearest output image channel within half a channel (if any). Otherwise, the chosen interpolation scheme will be used. Image channels that lie outside the MS frequency range or have no data mapped to them will be blank in the output image, but will be in the cube.

See the example in § F.1 for using the `'channel'` mode to image a spectral-line cube. In this case, we use:

```
mode      = 'channel'
nchan     =      46
start     =       5
width     =       1
```

which will produce a 46-channel cube starting with channel 5 of the MS with the same channel width as the MS. *Note: the `start` channel is in reference to the channels in the MS, not the subset selected by `spw`.*

5.2.5.3 Mode frequency

For `mode='frequency'`, an output image cube is created with `nchan` channels spaced evenly in frequency.

```
mode      = 'frequency' # Spectral image definition(mfs, channel, velocity,frequency)
nchan     =      -1    # Number of channels (planes) in output image
start     =          '' # Frequency of first image channel: e.g. '1.4GHz'(''=default)
width     =          '' # Image channel frequency width: e.g '1.0kHz'(''=default)
interpolation = 'linear' # Spectral interpolation(nearest, linear, cubic)
outframe  =          '' # velocity frame of output image
```

The frequency of the first output channel is given by `start` and spacing by `width`. Output channels have width also given by `width`. The sign of `width` determines whether the output channels ascend or descend in frequency. Data from the input MS with centers that lie within one-half an input channel overlap of the frequency range of $\pm \text{width}/2$ centered on the output channels are gridded together.

The defaults are designed to safely choose output cube channels to span the input MS(s). The default `nchan=-1` will choose the number of channels needed to span the frequencies of the channels in the MS. The defaults `start=''` and `width=''` will use the channel frequency and width of the first channel of the first specified spectral window selected in `spw`. **ALERT:** As in “channel” mode, this is currently the first channel of the first `spw`, not the first channel selected from that `spw`.

The `interpolation` sub-parameter (§ 5.2.5.5) sets how channels are gridded into the image cube planes.

Using the NGC5921 dataset as an example:

```
mode      = 'frequency'
nchan     =      21
start     = '1412.830MHz'
width     =   '50kHz'
outframe  =   'LSRK'
```

would produce a 21-channel output cube with 50 kHz wide channels rather than the default channelization of the MS (24.4 kHz).

5.2.5.4 Mode velocity

If `mode='velocity'` is chosen, then an output image cube with `nchan` channels will be created, with channels spaced evenly in velocity. Parameters are:

```
mode      = 'velocity' # Spectral image definition(mfs, channel, velocity,frequency)
nchan     =      -1 # Number of channels (planes) in output image
start     =      '' # Velocity of first image channel: e.g '0.0km/s'(''=default)
width     =      '' # Image channel velocity width: e.g '-1.0km/s'(''=default)
interpolation = 'linear' # Spectral interpolation(nearest, linear, cubic)
outframe  =      '' # velocity reference frame of output image; '' =input
veltype   =   'radio' # velocity definition
```

Note that velocities are calculated with respect to the rest frequency in the MS or specified through the `restfreq` parameter (§ 5.2.7).

The velocity of the first output channel is given by `start` and spacing by `width`. Averaging is as in `mode='frequency'`. The `interpolation` sub-parameter (§ 5.2.5.5) sets how channels are gridded into the image cube planes.

The defaults are designed to safely choose output cube channels to span the input MS(s). The default `nchan=-1` will choose the number of channels needed to span the velocities of the channels in the MS. The defaults `start=''` and `width=''` will use the channel velocity and width of the first channel of the first specified spectral window selected in `spw`. **ALERT:** As in “channel” mode, this is currently the first channel of the first `spw`, not the first channel selected from that `spw`.

Again, using the NGC5921 dataset as an example:

```
mode      = 'velocity'
nchan     =      21
start     = '1383.0km/s'
width     =   '10km/s'
outframe  =   'LSRK'
```

Note that in this case the velocity axis runs forward, as opposed to the default channelization for `'channel'` or `'frequency'`.

5.2.5.5 Sub-parameter interpolation

The `interpolation` sub-parameter controls how spectral channels in the MS are gridded into the output image cube. This is available in all modes except `'mfs'`. The options are: `'nearest'`, `'linear'`, `'cubic'`.

For `'nearest'`, the channels in `spw` beyond the first are mapped into the nearest output image channel within half a channel (if any).

For `'linear'`, the channels are gridded into the planes using weights given by a linear function of the frequency of the MS channel versus the plane. Each input channel will be mapped to 1 or 2 output planes. For most users, this is the best choice.

For `'cubic'`, the channels are gridded using a cubic interpolation function.

5.2.6 Parameter phasecenter

The `phasecenter` parameter indicates which of the field IDs should be used to define the phase center of the mosaic image, or what that phase center is in RA and Dec. The default action is to use the first one given in the `field` list.

For example:

```
phasecenter='5'                # field 5 in multi-src ms
phasecenter='J2000 19h30m00 -40d00m00' # specify position
```

Note that the format for angles prefers to use `hm` for RA/time units and `dm` for Dec/Angle units as separators. The colon `::` separator is interpreted as RA/time even if its used for the Dec, so be careful not to copy/paste from other sources.

5.2.7 Parameter restfreq

The value of the `restfreq` parameter, if set, will over-ride the rest frequency in the header of the first input MS to define the velocity frame of the output image.

ALERT: The `restfreq` parameter takes the options of transitions and frequencies as in the corresponding `plotxy` parameter (§ 3.4.2.12), but the frame information is controlled under the `mode` parameter (§ 5.2.5).

For example:

```
restfreq='115.2712GHz',
```

will set the rest frequency to that of the CO 1-0 line.

ALERT: Setting `restfreq` explicitly here in `clean` is good practice, and may be necessary if your MS has been concatenated from different files for different spectral windows (§ 2.2.9).

5.2.8 Parameter spw

The `spw` parameter selects the spectral windows that will be used to form the image, and possibly a subset of channels within these windows.

The `spw` parameter is a string with an integer, list of integers, or a range, e.g.

```
spw = '1'           # select spw 1
spw = '0,1,2,3'    # select spw 0,1,2,3
spw = '0~3'        # same thing using ranges
```

You can select channels in the same string with a `:` separator, for example

```
spw = '1:10~30'    # select channels 10-30 of spw 1
spw = '0:5~55,3:5;6;7' # chans 5-55 of spw 0 and 5,6,7 of spw 3
```

This uses the standard syntax for `spw` selection is given in § 2.3.3. See that section for more options.

Note that the order in which multiple `spws` are given is important for `mode = 'channel'`, as this defines the origin for the channelization of the resulting image.

5.2.9 Parameter stokes

The `stokes` parameter specifies the Stokes parameters for the resulting images. Note that forming Stokes `Q` and `U` images requires the presence of cross-hand polarizations (e.g. `RL` and `LR` for circularly polarized systems such as the VLA) in the data. Stokes `V` requires both parallel hands (`RR` and `:LL`) for circularly polarized systems or the cross-hands (`XY` and `YX`) for linearly polarized systems such as ALMA and ATCA.

This parameter is specified as a string of up to four letters (`IQUV`). For example,

```
stokes = 'I'       # Intensity only
stokes = 'IQU'     # Intensity and linear polarization
stokes = 'IV'      # Intensity and circular polarization
stokes = 'IQUV'    # All Stokes imaging
```

are common choices. () The output image will have planes (along the “polarization axis”) corresponding to the chosen Stokes parameters.

If as input to deconvolution tasks such as `clean`, the `stokes` parameter includes polarization planes other than `I`, then choosing `psfmode='hogbom'` (§ 5.3.1.2) or `psfmode='clarkstokes'` (§ 5.3.1.3) will clean (search for components) each plane sequentially, while `psfmode='clark'` (§ 5.3.1.1) will deconvolve jointly.

5.2.10 Parameter `uvtaper`

This controls the radial weighting of visibilities in the uv-plane (see § 5.2.11 below) through the multiplication of the visibilities by the Fourier transform of an elliptical Gaussian. This is itself a Gaussian, and thus the visibilities are “tapered” with weights decreasing as a function of uv-radius.

The `uvtaper` parameter expands the menu upon setting `uvtaper=True` to reveal the following sub-parameters:

```
uvtaper      =      True   # Apply additional uv tapering of visibilities.
  outertaper =          []   # uv-taper on outer baselines in uv-plane
  innertaper =          []   # uv-taper in center of uv-plane (not implemented)
```

The sub-parameters specify the size and optionally shape and orientation of this Gaussian in the uv-plane or optionally the sky plane. The `outertaper` refers to a Gaussian centered on the origin of the uv-plane.

Some examples:

```
outertaper=[]                # no outer taper applied
outertaper=['5klambda']      # circular uv taper FWHM=5 kilo-lambda
outertaper=['5klambda','3klambda','45.0deg'] # elliptical Gaussian
outertaper=['10arcsec']      # on-sky FWHM 10"
outertaper=['300.0']         # 300m in aperture plane
```

Note that if no units are given on the taper, then the default units are assumed to be meters in aperture plane.

ALERT: The `innertaper` option is not yet implemented.

5.2.11 Parameter weighting

In order to image your data, we must have a map from the visibilities to the image. Part of that map, which is effectively a convolution, is the weights by which each visibility is multiplied before gridding. The first factor in the weighting is the “noise” in that visibility, represented by the data weights in the MS (which is calibrated along with the visibility data). The weighting function can also depend upon the uv locus of that visibility (e.g. a “taper” to change resolution). This is actually controlled by the `uvtaper` parameter (see § 5.2.10). The weighting matrix also includes the convolution kernel that distributes that visibility onto the uv-plane during gridding before Fourier transforming to make the image of the sky. This depends upon the density of visibilities in the uv-plane (e.g. “natural”, “uniform”, “robust” weighting).

The user has control over all of these.

Inside the Toolkit:

The `im.weight` method has more weighting options than available in the imaging tasks. See the **User Reference Manual** for more information on imaging weights.

ALERT: You can find a weighting description in the online User Reference Manual at:

<http://casa.nrao.edu/docs/casaref/imager.weight.html>

The `weighting` parameter expands the menu to include various sub-parameters depending upon the mode chosen:

5.2.11.1 'natural' weighting

For `weighting='natural'`, visibilities are weighted only by the data weights, which are calculated during filling and calibration and should be equal to the inverse noise variance on that visibility. Imaging weight w_i of sample i is given by

$$w_i = \omega_i = \frac{1}{\sigma_k^2} \quad (5.1)$$

where the data weight ω_i is determined from σ_i is the rms noise on visibility i . When data is gridded into the same uv-cell for imaging, the weights are summed, and thus a higher uv density results in higher imaging weights. No sub-parameters are linked to this mode choice. It is the default imaging weight mode, and it should produce “optimum” image with with the lowest noise (highest signal-to-noise ratio). Note that this generally produces images with the poorest angular resolution, since the density of visibilities falls radially in the uv-plane

5.2.11.2 'uniform' weighting

For `weighting = 'uniform'`, the data weights are calculated as in 'natural' weighting. The data is then gridded to a number of cells in the uv-plane, and after all data is gridded the uv-cells are re-weighted to have “uniform” imaging weights. This pumps up the influence on the image of data with low weights (they are multiplied up to be the same as for the highest weighted data), which sharpens resolution and reduces the sidelobe level in the field-of-view, but increases the rms image noise. No sub-parameters are linked to this mode choice.

For uniform weighting, we first grid the inverse variance ω_i for all selected data onto a grid with uv cell-size given by $2/FOV$ where FOV is the specified field of view (defaults to the image field of view). This forms the gridded weights W_k . The weight of the i -th sample is then:

$$w_i = \frac{\omega_i}{W_k}. \quad (5.2)$$

5.2.11.3 'superuniform' weighting

The `weighting = 'superuniform'` mode is similar to the 'uniform' weighting mode but there is now an additional `npixels` sub-parameter that specifies a change to the number of cells on a side (with respect to uniform weighting) to define a uv-plane patch for the weighting renormalization. If `npixels=0` you get uniform weighting.

5.2.11.4 'radial' weighting

The `weighting = 'radial'` mode is a seldom-used option that increases the weight by the radius in the uv-plane, ie.

$$w_i = \omega_i \cdot \sqrt{u_i^2 + v_i^2}. \quad (5.3)$$

Technically, I would call that an inverse uv-taper since it depends on uv-coordinates and not on the data per-se. Its effect is to reduce the rms sidelobes for an east-west synthesis array. This option has limited utility.

5.2.11.5 'briggs' weighting

The `weighting = 'briggs'` mode is an implementation of the flexible weighting scheme developed by Dan Briggs in his PhD thesis. See:

<http://www.aoc.nrao.edu/dissertations/dbriggs/>

This choice brings up the sub-parameters:

```
weighting      = 'briggs' # Weighting to apply to visibilities
robust         = 0.0     # Briggs robustness parameter
npixels       = 0       # number of pixels to determine uv-cell size 0=> field of view
```

The actual weighting scheme used is:

$$w_i = \frac{\omega_i}{1 + W_k f^2} \quad (5.4)$$

where W_k is defined as in `uniform` and `superuniform` weighting, and

$$f^2 = \frac{(5 * 10^{-R})^2}{\frac{\sum_k W_k^2}{\sum_i \omega_i}} \quad (5.5)$$

and R is the robust parameter.

The key parameter is the `robust` parameter, which sets R in the Briggs equations. The scaling of R is such that $R = 0$ gives a good trade-off between resolution and sensitivity. The `robust` R takes value between -2.0 (close to uniform weighting) to 2.0 (close to natural).

Superuniform weighting can be combined with Briggs weighting using the `npixels` sub-parameter. This works as in `'superuniform'` weighting (§ 5.2.11.3).

5.2.11.6 'briggsabs' weighting

For `weighting='briggsabs'`, a slightly different Briggs weighting is used, with

$$w_i = \frac{\omega_i}{W_k R^2 + 2\sigma_R^2} \quad (5.6)$$

where R is the robust parameter and σ_R is the `noise` parameter.

This choice brings up the sub-parameters:

```

weighting      = 'briggsabs' # Weighting to apply to visibilities
  robust       =      0.0    # Briggs robustness parameter
  noise        = '0.0Jy'    # noise parameter for briggs weighting when rmode='abs'
  npixels      =      0      # number of pixels to determine uv-cell size 0=> field of view

```

Otherwise, this works as `weighting='briggs'` above (§ 5.2.11.5).

5.2.12 Parameter `vis`

The value of the `vis` parameter is either the name of a single MS, or a list of strings containing the names of multiple MSs, that should be processed to produce the image. The MS referred to by the first name in the list (if more than one) is used to determine properties of the image such as channelization and rest frequency.

ALERT!
Multi-MS handling is not percolated to the tasks yet, as we are still working on this. Use single MS only.

For example,

```
vis = 'ngc5921.ms'
```

set a single input MS, while

```
vis = ['ngc5921_day1.ms', 'ngc5921_day2.ms', 'ngc5921_day3.ms']
```

points to three separate measurement sets that will be gridded together to form the image. This means that you do not have to concatenate datasets, for example from different configurations, before imaging.

5.2.13 Primary beams in imaging

The CASA imaging task and tools use primary beams based on models for each observatory's antenna types. In addition to different antenna diameters, different functions may be used.

The voltage patterns are based on the following antenna primary beams, based on the `TELESCOPE_NAME` keyword in the `OBSERVATION` table:

VLA — Airy disk fitted to measurement. Note that a R/L beam squint is also included with feed dependent angle;

ALMA — Airy disk for 12m dish with a blockage of 1m;

ATA — Airy disk for 6m dish;

ATCA — polynomial fitted to measurement of main lobe;

BIMA, HATCREEK — Gaussian with halfwidth of $\lambda/2D$;

CARMA — Airy patterns for the BIMA or OVRO dish sizes as appropriate;

GBT — polynomial fitted to measurement of main lobe;

GMRT — VLA Airy disk scaled to 45.0m;

IRAMPDB — Airy disk for dish of 15m with a blockage of 1m;

NRAO12M — VLA beam scaled to 12m;

OVRO — VLA Airy disk scaled to 10.4m;

SMA — Spheroidal function fit to FWHM;

WSRT — polynomial fitted to measurement of main lobe;

If the telescope name is unknown, or is CARMA or ALMA, then the `DISH_DIAMETER` in the `ANTENNA` table is used with a scaled VLA pattern.

Note that for the purposes of mosaicing in `clean`, the primary beams that are Airy or spheroidal are best-behaved (see § 5.3.15).

5.3 Deconvolution using CLEAN (`clean`)

To create an image and then deconvolve it with the CLEAN algorithm, use the `clean` task. This task will work for single-field data, or for multi-field mosaics (§ 5.3.15), in both narrow and wide-field imaging modes. **ALERT:** There is now an experimental feature to image data taken with “heterogeneous” arrays with non-identical dish sizes (§ 5.3.16).

Toolkit Note: MEM is not included in `clean`, but is available in the toolkit.

The `clean` task uses many of the common imaging parameters. These are described above in § 5.2. There are also a number of parameters specific to `clean`. These are listed and described below.

The default inputs to `clean` are:

```
# clean :: Deconvolve an image with selected algorithm
vis          =      '' # name of input visibility file
imagenam    =      [''] # Pre-name of output images
outlierfile  =      '' # Text file with image names, sizes, centers
field       =      '' # Field Name
spw         =      '' # Spectral windows:channels: '' is all
selectdata  =      False # Other data selection parameters
mode        =      'mfs' # Type of selection (mfs, channel, velocity,frequency)
    nterms   =      1 # Number of taylor terms to use for modeling the sky frequency depend
    reffreq  =      '' # Reference frequency for MFS (relevant only if nterms > 1)

gridmode    =      '' # The kind gridding kernel to be used for FFT-based transforms
niter       =      500 # Maximum number of iterations
gain        =      0.1 # Loop gain for cleaning
```

```

threshold      = '0.0mJy' # Flux level to stop cleaning. Must include units
psfmode        = 'clark' # method of PSF calculation to use during minor cycles
imagermode     = ''     # Use csclean or mosaic. If '', use psfmode
multiscale     = []     # deconvolution scales (pixels); [] = default standard clean
interactive    = False  # use interactive clean (with GUI viewer)
mask           = []     # cleanbox(es), mask image(s), and/or region(s) used in cleaning
imsize        = [256, 256] # x and y image size in pixels, symmetric for single value
cell           = ['1.0arcsec', '1.0arcsec'] # x and y cell size. default unit arcsec
phasecenter    = ''     # Image phase center: position or field index
restfreq       = ''     # rest frequency to assign to image (see help)
stokes         = 'I'    # Stokes params to image (eg I,IV, QU,IQUV)
weighting      = 'natural' # Weighting of uv (natural, uniform, briggs, ...)
uvtaper        = False  # Apply additional uv tapering of visibilities.
modelimage     = ''     # Name of model image(s) to initialize cleaning
restoringbeam  = ['']   # Output Gaussian restoring beam for CLEAN image
pbcpr          = False  # Output primary beam-corrected image
minpb          = 0.1    # Minimum PB level to use
calready       = True   # Create scratch columns and store model visibilities for selfcal
async          = False  # If true the taskname must be started using clean(...)

```

The `clean` task will produce a number of output images based on the root name given in `imagename`. These include:

```

<imagename>.clean.image      # the restored image
<imagename>.clean.flux       # the effective response (e.g. for pbcpr)
<imagename>.clean.flux.pbcpr # the PB coverage (ftmachine='mosaic' only)
<imagename>.clean.model      # the model image
<imagename>.clean.residual   # the residual image
<imagename>.clean.psf        # the synthesized (dirty) beam

```

ALERT: The `<imagename>.clean.flux.pbcpr` image is new in Patch 4 version 2.4.0 and reflects the unweighted primary beam coverage (without weighting or gridding kernel factors) used for the `minpb` cutoff (§ 5.3.7). This is produced only for `imagermode='mosaic'` with `ftmachine='mosaic'`.

The mode, `psfmode`, `imagermode`, and `weighting` parameters open up other sub-parameters. These are detailed in the common imaging task parameters section (§ 5.2). The `gridmode` parameter (§ 5.3.13) is available to select more advanced imaging options such as widefield imaging and beam squint correction.

A typical setup for `clean` on the NGC5921 dataset, after setting parameter values, might look like:

```

vis           = 'ngc5921.usecase.ms.contsub' # Name of input visibility file
imagename     = 'ngc5921.usecase.clean' # Pre-name of output images
field         = '0' # Field Name
spw           = '' # Spectral windows:channels: '' is all
selectdata   = False # Other data selection parameters
mode          = 'channel' # Type of selection (mfs, channel, velocity, frequency)
  nchan       = 46 # Number of channels (planes) in output image
  start       = 5 # first input channel to use
  width       = 1 # Number of input channels to average

```

```

interpolation = 'linear' # Spectral interpolation (nearest, linear, cubic)

gridmode      =      '' # The kind gridding kernel to be used for FFT-based transforms
niter         =      6000 # Maximum number of iterations
gain          =      0.1 # Loop gain for cleaning
threshold     =      8.0 # Flux level to stop cleaning. Must include units
psfmode       =      'clark' # method of PSF calculation to use during minor cycles
imagermode    =      '' # Use csclean or mosaic, or image-plane only if ''
multiscale    =      [] # set deconvolution scales (pixels)
interactive   =      False # use interactive clean (with GUI viewer)
mask          =      [108, 108, 148, 148] # cleanbox(es), mask image(s), and/or region(s)
imsize        =      [256, 256] # x and y image size in pixels
cell          =      [15.0, 15.0] # x and y cell size. default unit arcsec
phasecenter   =      '' # Image phase center: position or field index
restfreq      =      '' # rest frequency to assign to image (see help)
stokes        =      'I' # Stokes params to image (eg I,IV, QU,IQUV)
weighting     =      'briggs' # Weighting to apply to visibilities
    robust    =      0.5 # Briggs robustness parameter
    npixels   =      0 # uv-cell size in pixels 0=> field of view

uvtaper       =      False # Apply additional uv tapering of visibilities.
modelimage    =      '' # Name of model image(s) to initialize cleaning
restoringbeam =      [''] # Output Gaussian restoring beam for CLEAN image
pbcor         =      False # Output primary beam-corrected image
minpb         =      0.1 # Minimum PB level to use
async         =      False

```

An example of the clean task to create a continuum image from many channels is given below:

```

clean(vis='ggtau.1mm.split.ms', # Use data in ggtau.1mm.split.ms
      imagename='ggtau.1mm',    # Name output images 'ggtau.1mm.*' on disk
      psfmode='clark',          # Use the Clark CLEAN algorithm
      imagermode='',            # Do not mosaic or use csclean
      mask='',                  # Do not use clean box or mask
      niter=500, gain=0.1,      # Iterate 500 times using gain of 0.1
      mode='mfs',               # multi-frequency synthesis (combine channels)
      spw='0~2:2~57',          # Combine channels from 3 spectral windows
      field='0',                #
      stokes='I',              # Image stokes I polarization
      weighting='briggs',       # Use Briggs robust weighting
      rmode='norm',robust=0.5,  # with robustness parameter of 0.5
      cell=[0.1,0.1],          # Using 0.1 arcsec pixels
      imsize=[256,256])        # Set image size = 256x256 pixels

```

This example will clean the entire inner quarter of the primary beam. However, if you want to limit the region over which you allow the algorithm to find clean components then you can make a deconvolution region (or mask). To use a deconvolution region, box, or mask, set the `mask` parameter (§ 5.3.6).

For example, you can set up a simple 'cleanbox' region. To do this, make a first cut at the image and clean the inner quarter. Then use the `viewer` to look at the image and get an idea of where the emission is located. You can use the `viewer adjustment` panel to view the image in pixel coordinates and read out the pixel locations of your cursor.

Inside the Toolkit:

The `im.clean` method is used for CLEANing data. There are a number of methods used to set up the clean, including `im.setoptions`.

Then, you can use those pixel read-outs you just go to define a clean box region where you specify the bottom-left-corner (blc) x & y and top-right-corner x& y locations. For example, say you have a continuum source near the center of your image between `blcx`, `blcy`, `trcx`, `trcy` = 80, 80, 120, 120. Then to use this region:

```
mask=[80,80,120,120]    # Set the deconvolution region as a simple box in the center.
```

The following are the `clean` specific parameters and their allowed values, followed by a description of carrying out interactive cleaning.

5.3.1 Parameter `psfmode`

The `psfmode` parameter chooses the “algorithm” that will be used to calculate the synthesized beam for use during the minor cycles in the image plane. The value types are strings. Allowed choices are 'clark' (default) and 'hogbom'.

5.3.1.1 The clark algorithm

In the 'clark' algorithm, the cleaning is split into minor and major cycles. In the minor cycles only the brightest points are cleaned, using a subset of the point spread function. In the major cycle, the points thus found are subtracted correctly by using an FFT-based convolution. This algorithm is reasonably fast. Also, for polarization imaging, Clark searches for the peak in $I^2 + Q^2 + U^2 + V^2$.

5.3.1.2 The hogbom algorithm

The `hogbom` algorithm is the “Classic” image-plane CLEAN, where model pixels are found iteratively by searching for the peak. Each point is subtracted from the full residual image using the shifted and scaled point spread function. In general, this is not a good choice for most imaging problems (`clark` or `csclean` are preferred) as it does not calculate the residuals accurately. But in some cases, with poor uv-coverage and/or a PSF with bad sidelobes, the Hogbom algorithm will do better as it uses a smaller beam patch. For polarization cleaning, Hogbom searches for clean peak in I , Q , U , and V independently.

5.3.1.3 The clarkstokes algorithm

In the 'clarkstokes' algorithm, the Clark psf (§ 5.3.1.1) is used, but for polarization imaging the Stokes planes are cleaned sequentially for components instead of jointly as in 'clark'. This means

that this is the same as 'clark' for Stokes I imaging only. This option can also be combined with `imagermode='csclean'` (§ 5.3.4).

5.3.2 The multiscale parameter

ALERT: The `multiscale` option is currently under development and should be used with caution and be considered as an “experimental” algorithm. The multi-scale CLEAN method is known to need careful tuning in order to properly converge. However, currently the only control for `multiscale` in the `clean` task is the setting of the `scales`.

To activate multi-scale mode, specify a non-blank list of scales in the `multiscale` parameter. e.g.

```
multiscale = [0,3,10,30]    # Four scales including point sources
```

These are given in numbers of pixels, and specify FWHM of the Gaussians used to compute the filtered images.

Setting the `multiscale` parameter to a non-empty list opens up the sub-parameter:

```
multiscale      = [0, 3, 10, 30]    # set deconvolution scales (pixels)
  negcomponent   =          -1      # Stop if largest scale finds this many neg components
```

The `negcomponent` sub-parameter is here to set the point at which the clean terminates because of negative components. For `negcomponent > 0`, component search will cease when this number of negative components are found at the largest scale. If `negcomponent = -1` then component search will continue even if the largest component is negative.

The CASA multi-scale algorithm uses “Multi-scale CLEAN” to deconvolve using delta-functions and circular Gaussians as the basis functions for the model, instead of just delta-functions or pixels as in the other clean algorithms. This algorithm is still in the experimental stage, mostly because we are working on better algorithms for setting the scales for the Gaussians. The sizes of the Gaussians are set using the `scales` sub-parameter.

We are working on defining a better algorithm for scale setting. In the toolkit, there is an `nscale` argument which sets scales

$$\theta_i = \theta_{bmin} 10^{(i-N/2)/2} \quad (5.7)$$

where $N = \text{nscales}$ and θ_{bmin} is the fitted FWHM of the minor axis of the CLEAN beam.

5.3.3 Parameter gain

The `gain` parameter sets the fraction of the flux density in the residual image that is removed and placed into the clean model at each minor cycle iteration. The default value is `gain = 0.1` and is suitable for a wide-range of imaging problems. Setting it to a smaller gain per cycle, such as `gain = 0.05`, can sometimes help when cleaning images with lots of diffuse emission. Larger values, up to `gain=1`, are probably too aggressive and are not recommended.

Inside the Toolkit:

The `im.setscales` method sets the multi-scale Gaussian widths. In addition to choosing a list of sizes in pixels, you can just pick a number of scales and get a geometric series of sizes.

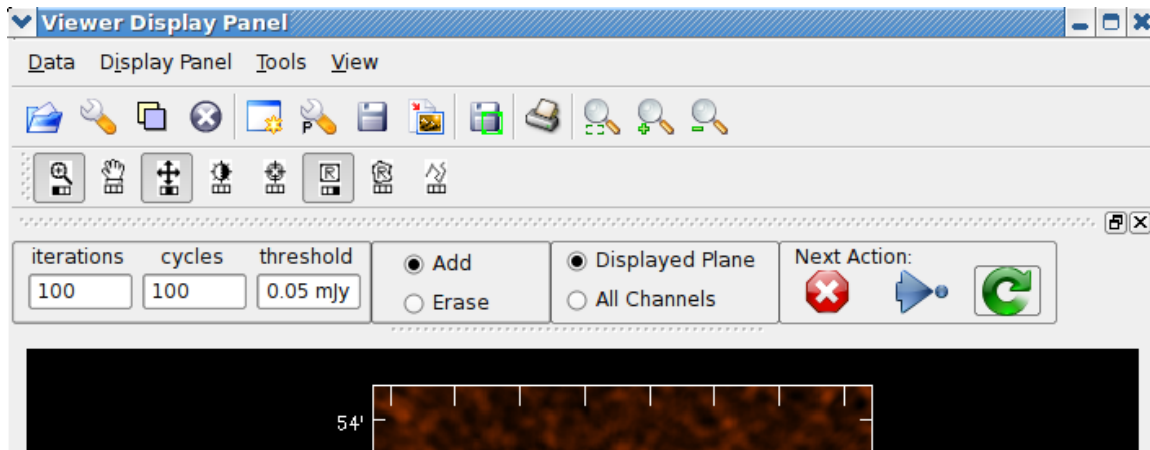


Figure 5.1: Close-up of the top of the interactive `clean` window. Note the boxes at the left (where the `iterations`, `cycles`, and `threshold` can be changed), the buttons that control add/erase, the application of mask to channels, and whether to stop, complete, or continue cleaning, and the row of Mouse-button tool assignment icons.

5.3.4 Parameter `imagermode`

This chooses the mode of operation of `clean`, either as single-field deconvolution using image-plane major and minor cycles only (`imagermode=''`), single-field deconvolution using Cotton-Schwab (CS) residual visibilities for major cycles (`imagermode='csclean'`), or multi-field mosaics using CS major cycles (`imagermode='mosaic'`).

In the default mode (`imagermode=''`), the major and minor clean cycles work off of the gridded FFT dirty image, with residuals updated using the PSF calculation algorithm set by the `psfmode` parameter (§ 5.3.1). This method is not recommended for high dynamic range or high fidelity imaging applications, but can be significantly faster than CS clean. Note that for this option only, if `mask=''` (no mask or box set) then it will clean the inner quarter of the image by default.

The `csclean` choice specifies the Cotton-Schwab algorithm. This opens up the sub-parameters

```
imagermode      = 'csclean' # Use csclean or mosaic. If '', use psfmode
  cyclefactor    =      1.5 # Change depth in between of csclean cycle
  cyclespeedup  =      -1  # Cycle threshold doubles in this number of iterations
```

These options are explained below. In the CS mode, cleaning is split into minor and major cycles. For each field, a minor cycle is performed using the PSF algorithm specified in `psfmode` (§ 5.3.1). At major-cycle breakpoints, the points thus found are subtracted from the original visibilities. A fast variant does a convolution using a FFT. This will be faster for large numbers of visibilities. If you want to be extra careful, double the image size from that used for the Clark clean and set a

mask to clean only the inner quarter or less (this is not done by default). This is probably the best choice for high-fidelity deconvolution of images without lots of large-scale structure.

Note that when using the Cotton-Schwab algorithm with a `threshold` (§ 5.3.12), there may be strange behavior when you hit the threshold with a major cycle. In particular, it may be above threshold again at the start of the next major cycle. This is particularly noticeable when cleaning a cube, where different channels will hit the threshold at different times.

ALERT: You will see a warning message in the logger, similar to this:

```
Zero Pixels selected with a Flux limit of 0.000551377 and a maximum Residual of 0.00751239
```

whenever it find 0 pixels above the threshold. This is normal, and not a problem, if you've specified a non-zero threshold. On the other hand, if you get this warning with the threshold set to the default of '0Jy', then you should look carefully at your inputs or your data, since this usually means that the masking is bad.

The option `imagermode='mosaic'` is for multi-field mosaics. This choice opens up the sub-parameters

```
imagermode      = 'mosaic' # Use csclean or mosaic. If '', use psfmode
mosweight       = False   # Individually weight the fields of the mosaic
ftmachine       = 'mosaic' # Gridding method for the image
scaletype      = 'SAULT'  # Controls scaling of pixels in the image plane.
cyclefactor     = 1.5     # change depth in between of csclean cycle
cyclespeedup   = -1      # Cycle threshold doubles in this number of iteration
```

These options are explained below.

5.3.4.1 Sub-parameter `cyclefactor`

This sub-parameter is activated for `imagermode='csclean'` and `'mosaic'`.

The `cyclefactor` parameter allows the user to change the threshold at which the deconvolution cycle will stop and then `degrid` and subtract the model from the visibilities to form the residual. This is with respect to the breaks between minor and major cycles that the `clean` part would normally force. Larger values force a major cycle more often.

This parameter in effect controls the threshold used by CLEAN to test whether a major cycle break and reconciliation occurs:

```
cycle threshold = cyclefactor * max sidelobe * max residual
```

Inside the Toolkit:

The `im.setmfcontrol` method sets the parameters that control the cycles and primary beam used in mosaicing.

If your uv-coverage results in a poor PSF, then you should reconcile often (a `cyclofactor` of 4 or 5); For reasonable PSFs, use `cyclofactor` in the range 1.5 to 2.0. For good PSFs, or for faster cleaning at the expense of some fidelity, we recommend trying a lower value, e.g. `cyclofactor = 0.25`, which at least in some of our mosaicing tests led to a speedup of around a factor of two with little change in residuals.

If `'mosaic'` or `'csclean'` diverges on your data, try a larger `cyclofactor`. A larger value typically increases the robustness of your deconvolution. The price, however, will be a slower algorithm.

5.3.4.2 Sub-parameter `cyclespeedup`

This sub-parameter is activated for `imagermode='csclean'` and `'mosaic'`.

The `cyclespeedup` parameter allows the user to let `clean` to raise the threshold at which a major cycle is forced if it is not converging to that threshold. To do this, set `cyclespeedup` to an integer number of iterations at which if the threshold is not reached, the threshold will be doubled. See `cyclofactor` above for more details. By default this is turned off (`cyclespeedup = -1`). In our tests, a value like `cyclespeedup = 50` has been used successfully.

5.3.4.3 Sub-parameter `ftmachine`

This sub-parameter is activated for `imagermode='mosaic'`. **Note:** The actual “ftmachine” used may be overridden by choices made to other parameters, such as `gridmode`.

The `ftmachine` parameter controls the gridding method and kernel to be used to make the image. A string value type is expected. Choices are: `'ft'`, `'sd'`, `'both'`, or `'mosaic'` (the default).

The `'ft'` option uses the standard gridding kernel (as used in `clean`).

The `'sd'` option forces gridding as in single-dish data.

For combining single-dish and interferometer MS in the imaging, the `'both'` option will allow `clean` to choose the `'ft'` or `'sd'` machines as appropriate for the data.

The `'mosaic'` option (the default) uses the Fourier transform of the primary beam (the aperture cross-correlation function in the uv-plane) as the gridding kernel. This allows the data from the multiple fields to be gridded down to a single uv-plane, with a significant speed-up in performance in most (non-memory limited) cases. The effect of this extra convolution is an additional multiplication (apodization) by the primary beam in the image plane. This can be corrected for, but does result in an image with optimal signal to noise ratio across it.

Inside the Toolkit:

The `im.setoptions` method sets the parameters relevant to mosaic imaging, such as the `ftmachine`.

The primary beams used in CASA are described in § 5.2.13.

ALERT: Note that making a non-square image (e.g. using unequal values in `imsize`) for `ftmachine='mosaic'` will grid the data into a uv-plane with correspondingly non-square cells. This has not been extensively tested, and may result in undesired image artifacts. We recommend that the user make square mosaic images when possible, but in principle non-square images should work.

5.3.4.4 Sub-parameter `mosweight`

If `mosweight=False` (default) then the data will be weighted for optimal signal to noise ratio across the mosaic image. This should be used for most cases.

If `mosweight=True` then individual mosaic field relative weights will be readjusted on a per visibility basis (much like uniform gridding weights). This may give better performance in cases where one or a few fields in the mosaic have drastically different weights and/or integration time, and it is desired that the noise be more “uniform” across the mosaic image. Use this with care, we have not explored its use fully.

5.3.4.5 Sub-parameter `scaletype`

The `scaletype` parameter controls weighting of pixels in the image plane. This sub-parameter is activated for `imagermode='mosaic'`.

The default `scaletype='PBCOR'` scales the image to have the correct flux scale across it (out to the beam level cutoff `minpb`). This means that the noise level will vary across the image, being increased by the inverse of the weighted primary beam responses that are used to rescale the fluxes.

This option should be used with care, particularly if your data has very different exposure times (and hence intrinsic noise levels) between the mosaic fields.

If `scaletype='SAULT'` then the image will be scaled so as to have constant noise across it. This means that the point source response function varies across the image attenuated by the weighted primary beam(s). However, this response is output in the `.flux` image and can be later used to correct this.

Note that this scaling as a function of position in the image occurs after the weighting of mosaic fields specified by `mosweight` and implied by the gridding weights (`ftmachine` and `weighting`).

5.3.4.6 The threshold revisited

For mosaics, the specification of the threshold is not straightforward, as it is in the single field case. This is because the different fields can be observed to different depths, and get different weights in the mosaic. We now provide internal rescaling (based on `scaletype`) so `clean` does its component search on a properly weighted and scaled version of the sky.

Inside the Toolkit:

The `im.setmfcontrol` method gives more options for controlling the primary beam and noise across the image.

For `ftmachine='ft'`, the minor cycles of the deconvolution are performed on an image that has been weighted to have constant noise, as in 'SAULT' weighting (see § 5.3.4.5). This is equivalent to making a dirty mosaic by coadding dirty images made from the individual pointings with a sum of the mosaic contributions to a given pixel weighted by so as to give constant noise across the image. This means that the flux scale can vary across the mosaic depending on the effective noise (higher weighted regions have lower noise, and thus will have higher “fluxes” in the 'SAULT' map). Effectively, the flux scale that threshold applies to is that at the center of the highest-weighted mosaic field, with higher-noise regions down-scaled accordingly. Compared to the true sky, this image has a factor of the PB, plus a scaling map (returned in the `.flux` image). You will preferentially find components in the low-noise regions near mosaic centers.

When `ftmachine='mosaic'` and `scaletype='SAULT'`, the deconvolution is also performed on a “constant noise image”, as detailed above for 'ft'.

ALERT: The intrinsic image made using `ftmachine='mosaic'` is equivalent to a dirty mosaic that is formed by coadding dirty images made from the individual fields after apodizing each by the PB function. Thus compared to the true sky, this has a factor of the PB^2 in it. You would thus preferentially find components in the centers of the mosaic fields (even more so than in the 'ft' mosaics). We now rescale this image internally at major-cycle (and interactive) boundaries based on `scaletype`, and do not have a way to clean on the raw unscaled dirty image (as was done in previous released versions).

5.3.5 Parameter interactive

If `interactive=True` is set, then an interactive window will appear at various “cycle” stages while you clean, so you can set and change mask regions. These breakpoints are controlled by the `npercyc` sub-parameter which sets the number of iterations of clean before stopping.

```
interactive    =   True    # use interactive clean (with GUI viewer)
npercyc       =    100    # Number of iterations before interactive prompt
```

Note that `npercyc` is currently the only way to control the breakpoints in interactive clean.

For spectral cube imaging, it is often easier to deal with each channel in turn, rather than cleaning all channels in each cycle. We therefore provide the `chaniter=True` option under 'mode', where it will clean a channel fully before moving to the next one. You will set masks for each channel.

See the example of interactive cleaning in § 5.3.14.

5.3.6 Parameter mask

The `mask` parameter takes a list of elements, each of which can be a list of coordinates specifying a box, or a string pointing to the name of a cleanbox file, mask image, or region file. These are used by CLEAN to define a region to search for components.

Note that for `imagermode=''` (§ 5.3.4) the default with `mask=''` is to restrict clean to the inner quarter of the image.

5.3.6.1 Setting clean boxes

If `mask` is given a list, these are taken to be pixel coordinates for the `b1c` and `trc` (bottom-left and top-right corners) of one or more rectangular boxes. Each box is a four element list. For example,

```
mask = [ [110,110,150,145], [180,70,190,80] ]
```

defines two boxes.

5.3.6.2 Using clean box files

You can provide `mask` a string with the name of an ASCII file containing the BLC, TRC of the boxes with one box per line. Each line should contain five numbers

```
<fieldindex> <b1c-x> <b1c-y> <trc-x> <trc-y>
```

with whitespace separators. Currently the `<fieldindex>` is ignored.

Here is an example cleanbox file:

```
CASA <21>: !cat mycleanbox.txt
IPython system call: cat mycleanbox.txt
1 108 108 148 148
2 160 160 180 180
```

NOTE: In future patches we will include options for the specification of circular and polygonal regions in the `cleanbox` file, as well as the use of world coordinates (not just pixel) and control of plane ranges for the boxes. For now, use the `mask` mechanism for more complicated CLEAN regions.

5.3.6.3 Using clean mask images

You can give the `mask` parameter a string containing the name of a mask image to be used for CLEAN to search for components. You can use the `makemask` task (§ 5.5.2) to construct this mask, or use one made using `interactive=True` (§ 5.3.5).

5.3.6.4 Using region files

You can give the `mask` parameter a string pointing to a file that describes a *region*. This region file can be generated in the `viewer` (§ 7) or in the `boxit` task (§ 5.5.1) from an image.

5.3.7 Parameter `minpb`

The `minpb` parameter sets the level down to which the primary beam (or more correctly the voltage patterns in the array) can go and have a given pixel included in the image. This is important as it defines where the edge of the visible image or mosaic is. The default is 0.1 or equivalent to the 10% response level. If there is a lot of emission near the edge, then set this lower if you want to be able to clean it out.

NOTE: The `minpb` parameter is the level in the “primary beam” (PB) at which the cut is made. If you are using `ftmachine='mosaic'` (§ 5.3.4.3), this will show up in the `.flux.pbcoverage` image (new in version 2.4.0). See the discussion of `threshold` (§ 5.3.4.6) for related issues.

5.3.8 Parameter `modelimage`

The `modelimage` parameter specifies the name(s) of one or more input starting image(s) to use to calculate the first residual before cleaning. These are used in addition to any image with a name defaulting from the `imagename` root (e.g. on a restart). The output model will contain this model plus clean components found during deconvolution.

If the units of the image are `Jy/pixel`, then this is treated as a model image.

If the units of the image are `Jy/beam` or Jy per solid angle, then this is treated as a “single-dish” image and rescaled by the resolution (in the `'beam'` image header keyword). Inclusion of the SD image here is superior to feathering it in later. See § 5.4 for more information on feathering.

5.3.9 Parameter `niter`

The `niter` parameter sets the maximum total number of minor-cycle CLEAN iterations to be performed during this run of `clean`. If restarting from a previous state, it will carry on from where it was. Note that the `threshold` parameter can cause the CLEAN to be terminated before the requested number of iterations is reached.

5.3.10 Parameter `pbcor`

The `pbcor` parameter controls whether the final `.image` is scaled to correct for the Primary Beam of the array or not.

If `pbcor=False` (the default), then no such scaling is done and the image is in whatever “raw” scaling used by the `imagermode` algorithm underneath. For single-field cleaning with `imagermode=''` or `'csclean'`, this is the standard constant-noise image. If `imagermode='mosaic'`, then this is the `'SAULT'` scaled image (regardless of what `scaletype` is set to).

If `pbcor=True`, then at the end of deconvolution and imaging the “raw” image is rescaled by dividing by the noise and PB correction image. This is what is output by `clean` as the `.flux` image.

Note that regardless of what you set `pbcor` to, you can recover the other option using `immath` (§ 6.5) to either multiply or divide by the `.flux` image.

5.3.11 Parameter `restoringbeam`

The `restoringbeam` parameter allows the user to set a specific Gaussian restoring beam to make the final restored `.image` from the final `.model` and residuals.

If `restoringbeam=''` (the default), then the restoring beam is calculated by fitting to the PSF (e.g. the `.psf` image). For a mosaic, this is at the center of the field closest to the `phasecenter`.

To specify a restoring beam, provide `restoringbeam` a list of [`bmaj`, `bmin`, `bpa`] which are the parameters of an elliptical Gaussian. The default units are in arc-seconds for `bmaj`, `bmin` components and degrees for the `bpa` component.

For example,

```
restoringbeam=['10arcsec']           # circular Gaussian FWHM 10"
restoringbeam=['10.0','5.0','45.0deg'] # 10"x5" at PA=45 degrees
```

5.3.12 Parameter `threshold`

The `threshold` parameter instructs `clean` to terminate when the maximum (absolute?) residual reaches this level or below. Note that it may not reach this residual level due to the value of the `niter` parameter which may cause it to terminate early.

If `threshold` is given a floating-point number, then this is the threshold in milli-Jansky.

You can also supply a flux density *quanta* to `threshold`, e.g.

```
threshold = '8.0mJy'
threshold = '0.008Jy'
```

(these do the same thing).

5.3.13 Parameter `gridmode`

The `gridmode` parameter is now provided to access more advanced deconvolution capabilities. The default `gridmode=''` is recommended for most cases.

The `gridmode='widefield'` option allows imaging in the wide-field regime where the W-term is not negligible. The CASA implementation allows both standard uv-plane faceting as well as the W-Projection algorithm² or a combination of the two. Its sub-parameters are:

```
gridmode      = 'widefield' # Gridding kernel for FFT-based transforms, default='' None
  wprojplanes =           1 # Number of w-projection planes for convolution
  facets      =           1 # Number of facets along each axis (main image only)
```

²Cornwell et al. IEEE JSTSP (2008).

The `wprojplanes` parameter sets the number of pre-computed w-planes used for the W-Projection algorithm (`wprojplanes=1` disables w-projection). The `facets` parameter sets the number of facets used. W-Projection, if used, is done for each facet. See § 5.3.18 below for more on wide-field imaging.

The `gridmode='aprojection'` option enables the gridding-based correction for RL beam-squint in off-axis systems (such as the VLA). The sub-parameters are:

```
gridmode   = 'aprojection' # Gridding kernel for FFT-based transforms, default='' None
  cfcache  = 'cfcache.dir' # Convolution function cache directory
  painc    =      360.0     # Parallax angle increment (degrees) for convolution function
```

Higher dynamic range imaging applications will require a-projection with correspondingly small (e.g. 10-degree) `painc` at the cost of speed. **ALERT:** This option is still under test and development, so be careful in its use.

5.3.14 Interactive Cleaning — Example

If `interactive=True` is set, then an interactive window will appear at various “cycle” stages while you clean, so you can set and change mask regions. These breakpoints are controlled by the `npercycle` sub-parameter which sets the number of iterations of clean before stopping.

The window controls are fairly self-explanatory. It is basically a form of the `viewer`. A close-up of the controls are shown in Figure 5.1, and an example can be found in Figures 5.2–5.4. You assign one of the drawing functions (rectangle or polygon, default is rectangle) to the right-mouse button (usually), then use it to mark out regions on the image. Zoom in if necessary (standard with the left-mouse button assignment). Double-click inside the marked region to add it to the mask. If you want to reduce the mask, click the **Erase** radio button (rather than **Add**), then mark and select as normal. When finished setting or changing your mask, click the green clockwise arrow “Continue Cleaning” Next Action button. If you want to finish your clean with no more changes to the mask, hit the blue right arrow “Apply mask edits and proceed with non-interactive clean” button. If you want to terminate the clean, click the red X “Stop deconvolving now” button.

While stopped in an interactive step, you can change a number of control parameters in the boxes provided at the left of the menu bar. The main use of this is to control how many `iterations` before the next breakpoint (initially set to `npercycle`), how many cycles before completion (initially equal to `niter/npercycle`), and to change the `threshold` for ending cleaning. Typically, the user would start with a relatively small number of iterations (50 or 100) to clean the bright emission in tight mask regions, and then increase this as you get deeper and the masking covers more of the emission region. For extended sources, you may end up needing to clean a large number of components (10000 or more) and thus it is useful to set `niter` to a large number to begin with — you can always terminate the clean interactively when you think it is done. Note that if you change `iterations` you may also want to change `cycles` or your clean may terminate before you expect it to.

For strangely shaped emission regions, you may find using the polygon region marking tool (the second from the right in the button assignment toolbar) the most useful.

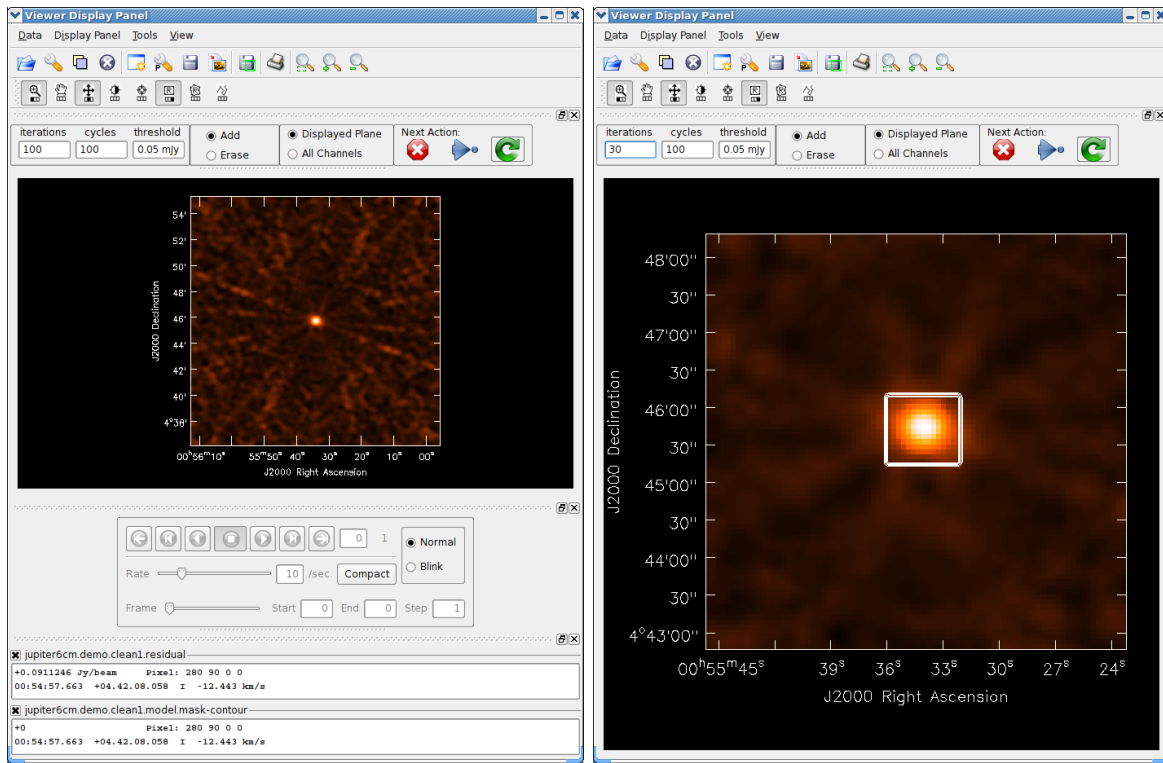


Figure 5.2: Screen-shots of the interactive `clean` window during deconvolution of the VLA 6m Jupiter dataset. We start from the calibrated data, but before any self-calibration. In the initial stage (left), the window pops up and you can see it dominated by a bright source in the center. Next (right), we zoom in and draw a box around this emission. We have also at this stage dismissed the tape deck and Position Tracking parts of the display (§ 7.2.1) as they are not used here. We have also changed the `iterations` to 30 for this boxed clean. We will now hit the Next Action **Continue Cleaning** button (the green clockwise arrow) to start cleaning.

See the example of cleaning and self-calibrating the Jupiter 6cm continuum data given below in Appendix F.2. The sequence of cleaning starting with the “raw” externally calibrated data is shown in Figures 5.2 – 5.4.

The final result of all this cleaning for Jupiter is shown in Figure 5.5. The `viewer` (§ 7) was used to overplot the polarized intensity contours and linear polarization vectors calculated using `immath` (§ 6.5) on the total intensity. See the following chapters on how to make the most of your imaging results.

For spectral cube images you can use the tapedeck to move through the channels. You also use the panel with radio buttons for choosing whether the mask you draw applies to the `Displayed Plane` or to `All Channels`. See Figure 5.6 for an example. Note that currently the `Displayed Plane` option is set by default. This toggle is unimportant for single-channel images or `mode='mfs'`.

Advanced Tip: Note that while in interactive `clean`, you are using the `viewer`. Thus, you have

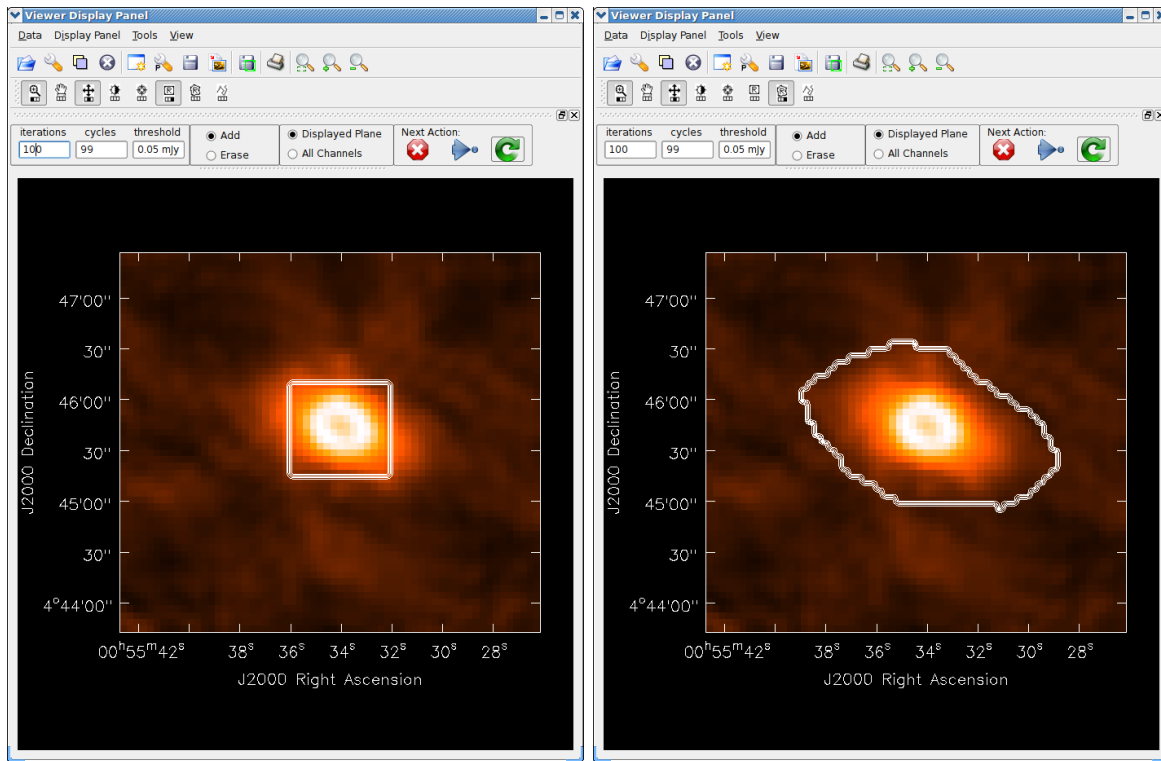


Figure 5.3: We continue in our interactive cleaning of Jupiter from where Figure 5.2 left off. In the first (left) panel, we have cleaned 30 iterations in the region previously marked, and are zoomed in again ready to extend the mask to pick up the newly revealed emission. Next (right), we have used the Polygon tool to redraw the mask around the emission, and are ready to **Continue Cleaning** for another 100 iterations.

the ability to open and register other images in order to help you set up the clean mask. For example, if you have a previously cleaned image of a complex source or mosaic that you wish to use to guide the placement of boxes or polygons, just use the **Open** button or menu item to bring in that image, which will be visible and registered on top of your dirty residual image that you are cleaning on. You can then draw masks as usual, which will be stored in the mask layer as before. Note you can blink between the new and dirty image, change the colormap and/or contrast, and carry out other standard viewer operations. See § 7 for more on the use of the viewer.

ALERT: Currently, interactive spectral line cleaning is done globally over the cube, with halts for interaction after searching all channels for the requested `npercycle` total iterations. It is more convenient for the user to treat the channels in order, cleaning each in turn before moving on. This will be implemented in an upcoming update.

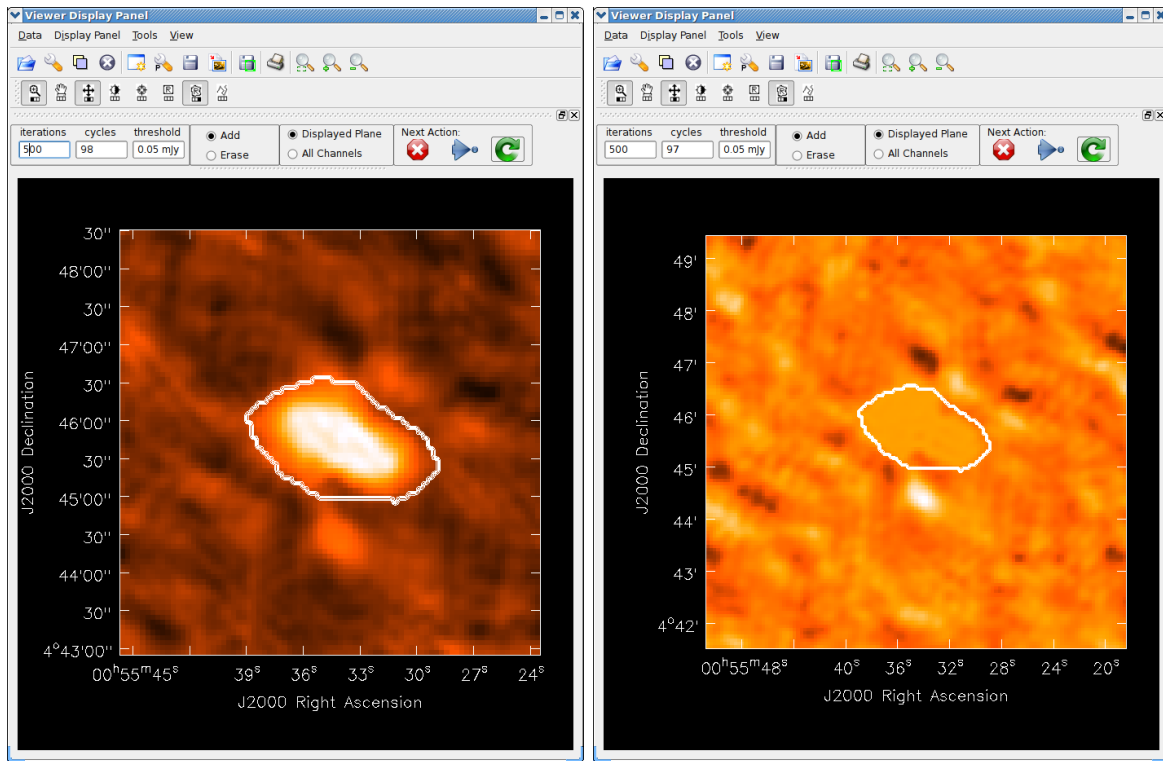


Figure 5.4: We continue in our interactive cleaning of Jupiter from where Figure 5.3 left off. In the first (left) panel, it has cleaned deeper, and we come back and zoom in to see that our current mask is good and we should clean further. We change `npercycle` to 500 (from 100) in the box at upper right of the window. In the final panel (right), we see the results after this clean. The residuals are such that we should terminate the `clean` using the red X button and use our model for self-calibration.

5.3.15 Mosaic imaging

The `clean` task contains the capability to image multiple pointing centers together into a single “mosaic” image. This ability is controlled by setting `imagermode='mosaic'` (§ 5.3.4).

The key parameter that controls how `clean` produces the mosaic is the `ftmachine` sub-parameter (§ 5.3.4.3). For `ftmachine='ft'`, `clean` will perform a weighted combination of the images produced by transforming each mosaic pointing separately. This can be slow, as the individual sub-images must be recombined in the image plane. **NOTE:** this option is preferred for data taken with sub-optimal mosaic sampling (e.g. fields too far apart, on a sparse irregular pattern, etc.).

The primary beams used in CASA are described in § 5.2.13.

If `ftmachine='mosaic'`, then the data are gridded onto a single uv-plane which is then transformed to produce the single output image. This is accomplished by using a gridding kernel that approximates the transform of the primary beam pattern. Note that for this mode the

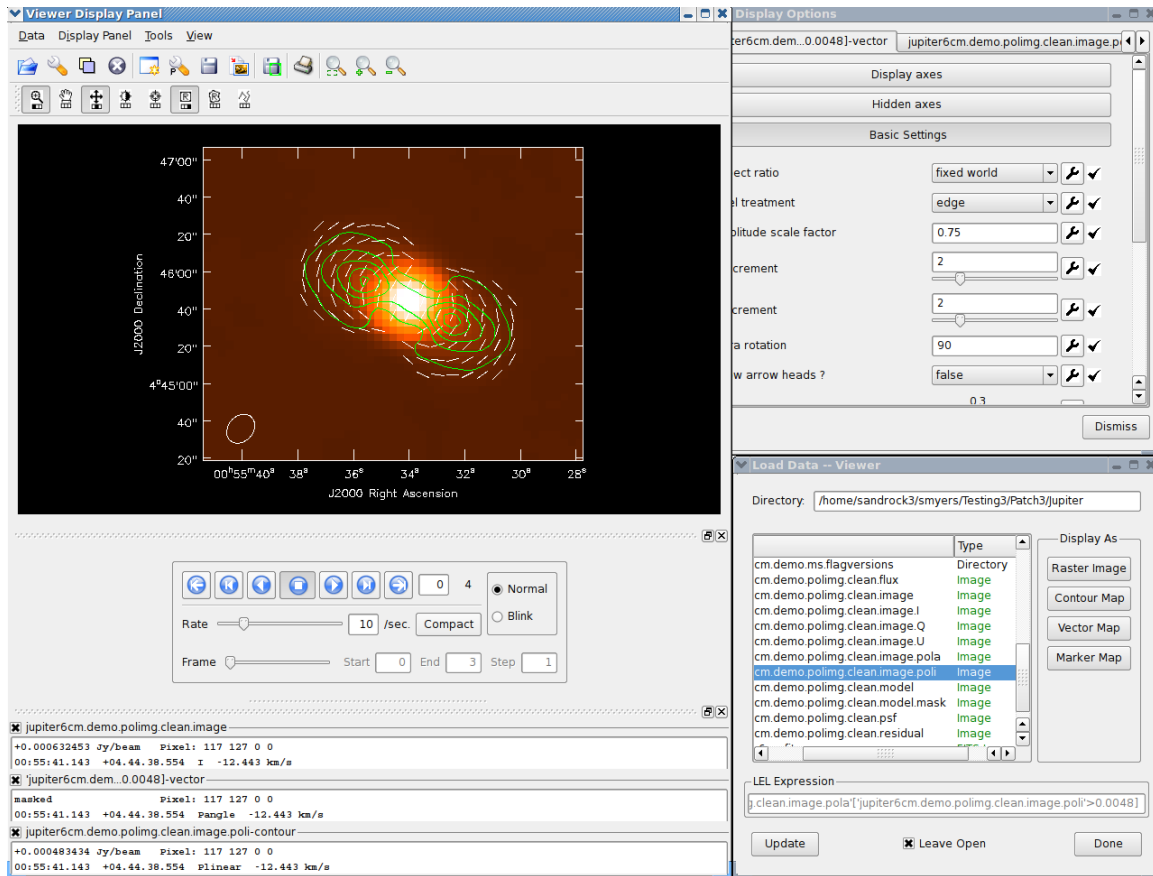


Figure 5.5: After clean and self-calibration using the intensity image, we arrive at the final polarization image of Jupiter. Shown in the viewer superimposed on the intensity raster is the linear polarization intensity (green contours) and linear polarization B-vectors (vectors). The color of the contours and the sampling and rotation by 90 degrees of the vectors was set in the Display Options panel. A LEL expression was used in the Load Data panel to mask the vectors on the polarized intensity.

`<imagename>.flux` image includes this convolution kernel in its effective weighted response pattern (needed to “primary-beam correct” the output image). For this mode only, an additional image `<imagename>.flux.pbcoverage` is produced that is the primary-beam coverage only used to compute the `minpb` cutoff (§ 5.3.7).

ALERT: In order to avoid aliasing artifacts for `ftmachine='mosaic'` in the mosaic image, due to the discrete sampling of the mosaic pattern on the sky, you should make an image in which the desired unmasked part of the image (above `minpb`) lies within the inner quarter. In other words, make an image twice as big as necessary to encompass the mosaic.

It is also important to choose an appropriate `phasecenter` for your output mosaic image (§ section:im.pars.phasecenter).

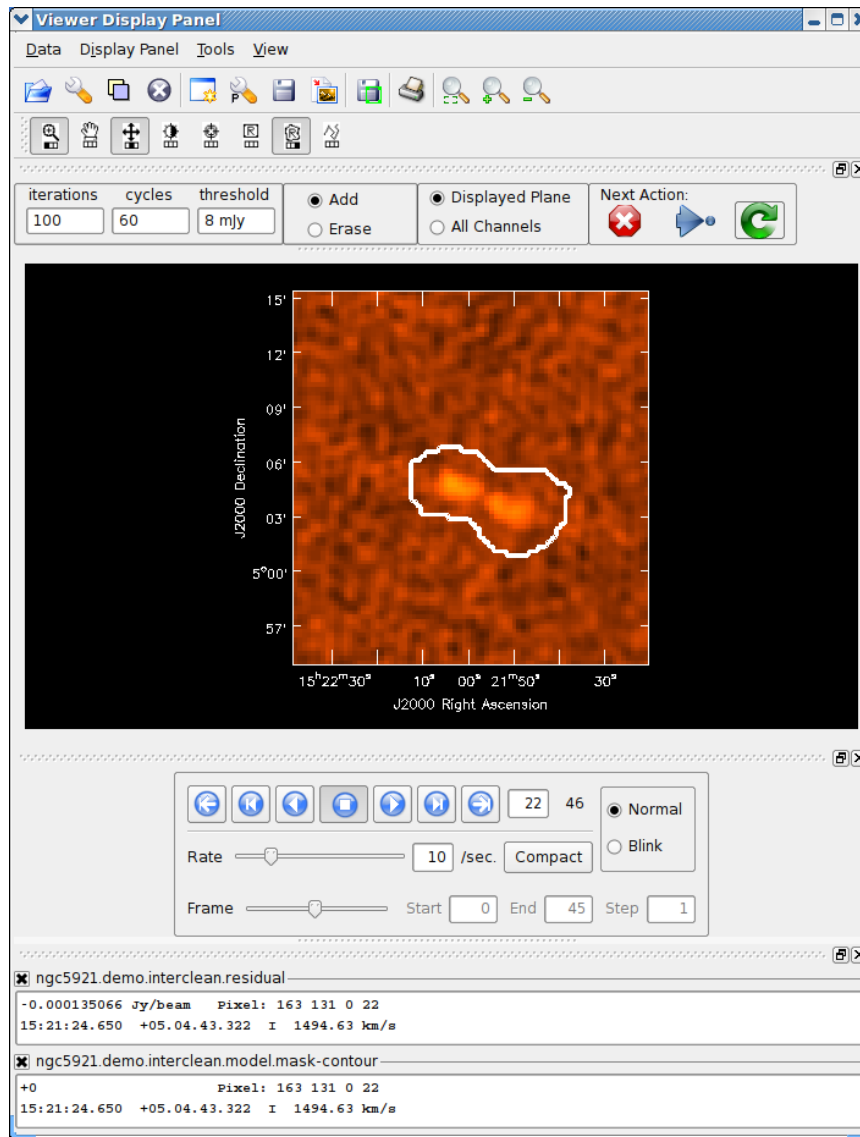


Figure 5.6: Screen-shot of the interactive `clean` window during deconvolution of the NGC5921 spectral line dataset. Note where we have selected the mask to apply to the `Displayed Plane` rather than `All Channels`. We have just used the Polygon tool to draw a mask region around the emission in this channel, which will apply to this channel only.

An example of a simple mosaic `clean` call is shown below:

```
clean(vis='n4826_tboth.ms',
      imagename='tmosaic',
      mode='channel',
      nchan=30,start=46,           # Make the output cube 30 chan
```

```

width=4,                # start with 46 of spw 0, avg by 4 chans
spw='0~2',
field='0~6',
cell=[1.,1.],
imsize=[256,256],
stokes='I',
psfmode='clark',
niter=500,
imagermode='mosaic',
scaletype='SAULT',
cyclefactor=0.1)

```

Another example of mosaic imaging, this time using 3mm BIMA data, is given in Appendix F.3.

5.3.16 Heterogeneous imaging

The `clean` task and underlying tools can now handle cases where there are multiple dish sizes, and thus voltage patterns and primary beams, in the array. This is effected by using the dish sizes stored in the `ANTENNA` sub-table of the MS. Depending on how the data was written and imported into CASA, the user may have to manually edit this table to insert the correct dish sizes (e.g. using `browsetable` or the `tb` table tool).

ALERT: This feature is new in Patch 3, and has not been extensively tested. Currently, this works only for an MS where the `OBSERVATORY` keyword is `CARMA`, `ALMA`, or is unknown. You must set `imagermode='mosaic'` with `ftmachine='mosaic'`, even when imaging a single field, to use this feature. This will be improved, and made easier to set and use, in future releases.

5.3.17 Polarization imaging

The `clean` task handles full and partial Stokes polarization imaging through the setting of the `stokes` parameter (§ 5.2.9). The subsequent deconvolution of the polarization planes of the image and the search for clean components is controlled by the `psfmode` parameter (§ 5.3.1). If the `stokes` parameter includes polarization planes other than I, then choosing `psfmode='hogbom'` (§ 5.3.1.2) or `psfmode='clarkstokes'` (§ 5.3.1.3) will clean (search for components) each plane sequentially, while `psfmode='clark'` (§ 5.3.1.1) will deconvolve jointly.

The interactive `clean` example given above (§ 5.3.14) shows a case of polarization imaging. You can also find the script for this example in Appendix F.2.

5.3.18 Wide-field imaging and deconvolution in `clean`

When imaging sufficiently large angular regions, the sky can no longer be treated as a two-dimensional plane and the use of the standard `clean` task will produce distortions around sources that become increasingly severe with increasing distance from the phase center. In this case, one must use a “wide-field” imaging algorithm such as w-projection or faceting.

When is wide-field imaging needed? It depends on the expected dynamic range the image. In order to keep the phase error associated with the sky/array curvature less than about 5° (good to about 500:1 dynamic range), use the following table, suitably scaled, for guidance:

Maximum Radius of Image Before widefield is Needed
Assuming 5 deg phase error and 35-km Baseline

Wavelength	Radius of image
6 cm	1.4 arcmin
20 cm	2.6 arcmin
90 cm	5.3 arcmin

Radius of image \sim SQRT (Wavelength * phase error / Maximum baseline)
(arcmin) (cm) (deg) (km)

If a relatively small image is being made, but there are outliers sources beyond the above limits, then widefield should also be used.

The relevant inputs for `clean` for wide-field imaging are:

```
gridmode      = 'widefield'      # The kind gridding kernel to be used for FFT-based transforms
wprojplanes   =          1       # Number of w-projection planes for convolution
facets        =          1       # Number of facets along each axis (main image only)
```

Most of the `clean` parameters behave as described previously.

Wide-field imaging can be carried out using two major modes: First, the w-projection mode as chosen with `ftmachine` deals with the w-term (the phase associated with the sky/array curvature) internally. Secondly, the image can be broken into many facets, each small enough so that the w-term is not significant. These two basic methods can be combined, as discussed below in § 5.3.18.4.

5.3.18.1 Outlier fields

When using wide-field imaging, the position and image size of any independent images must be specified. This can be done in two ways. The first method uses input from a text file `outlierfile`. For example:

```
vis           = 'wfield.ms'      # name of input visibility file
imagenam     = 'wf'             # Pre-name of output images
outlierfile   = 'setup.txt'     # Text file with image names, sizes, centers
imsize       = [256, 256]      # Image size in pixels (nx,ny), symmetric for single value
cell         = '1.0arcsec'     # The image cell size in arcseconds [x,y].
phasecenter  = ''              # Field Identifier or direction of the image phase center
```

The `cell` is identical for all fields, and `imsize` and `phasecenter` are not used. The output image names will be `wf.main`, `wf.out1`, etc.

The file `outlierfile="setup.txt"` is an AIPS-style field file used to locate outlier fields. For example:

```

C  main 2048 2048   13 27 20.98   43 26 28.0   # Main field with image size and phase center
C  out1  128  128   13 30 52.158   43 23 08.00   # First outlier field specification
                                     etc

```

will make a big main image with outliers. The C in column 1 must be present, although it not presently used.

The alternative input, needing no additional outlier file, versus `imsize` plus `phasecenter`: is:

```

vis           = 'wfield.ms'      # name of input visibility file
imagename    = 'wf'             # Pre-name of output images
outlierfile  = 'image_setup.txt' # Text file with image names, sizes, centers
imsize       = [2048, 2048, 128, 128] # Image size in pixels (nx,ny)
cell         = '1.0arcsec'      # The image cell size in arcseconds [x,y].
phasecenter  = ['J2000 13h27m20.98 43d26m28.0', 'J2000 13h30m52.158 43d23m08.00']

```

5.3.18.2 Setting up w-projection

The w-projection mode is controlled using `wprojplanes` sub-parameter, e.g.

```

gridmode      = 'widefield'      # The kind gridding kernel to be used for FFT-based transforms
wprojplanes   =          64      # Number of w-projection planes for convolution
facets        =           1      # Number of facets along each axis (main image only)

```

will construct 64 w-projection planes.

A reasonable value for `wprojplanes` is equal to $n_w = B_{max}(\text{ink}\lambda) \times \text{imagewidth}(\text{inarcmin}^2)/600$, with a minimum value of $n_w = 16$. The w-projection algorithm is much faster than using faceting, but it does consume a lot of memory. On most 32-bit machines with 1 or 2 Mbytes of memory, images larger than about 4000×4000 cannot be made.

5.3.18.3 Setting up faceting

Faceting will break the image into many small parts. This is invoked using `facets`:

```

gridmode      = 'widefield'      # The kind gridding kernel to be used for FFT-based transforms
wprojplanes   =           1      # Number of w-projection planes for convolution
facets        =           7      # Number of facets along each axis (main image only)

```

In this example the image is broken into 49 (7×7) facets.

A reasonable value of facets is such that the image width of each facet does not need the w-term correction. The computation method with pure faceting is slow, so that w-projection is recommended

5.3.18.4 Combination of w-projection and faceting

You can also use a combination of w-projection and faceting:

```
gridmode          = 'widefield'      # The kind gridding kernel to be used for FFT-based transforms
  wprojplanes     =          32      # Number of w-projection planes for convolution
  facets         =           3      # Number of facets along each axis (main image only)
```

This hybrid method allows for a smaller number of `wprojplanes` in order to try to conserve memory if the image size approached the memory limit of the computer. However, there is a large penalty in execution time.

5.4 Combined Single Dish and Interferometric Imaging (feather)

The term “feathering” is used in radio imaging to describe how to combine or “feather” two images together by forming a weighted sum of their Fourier transforms in the (gridded) uv -plane. Intermediate size scales are down-weighted to give interferometer resolution while preserving single-dish total flux density.

The feathering technique does the following:

1. The single-dish and interferometer images are Fourier transformed.
2. The beam from the single-dish image is Fourier transformed ($FTSDB(u, v)$).
3. The Fourier transform of the interferometer image is multiplied by $(1 - FTSDB(u, v))$. This basically down weights the shorter spacing data from the interferometer image.
4. The Fourier transform of the single-dish image is scaled by the volume ratio of the interferometer restoring beam to the single dish beam.
5. The results from 3 and 4 are added and Fourier transformed back to the image plane.

The term feathering derives from the tapering or down-weighting of the data in this technique; the overlapping, shorter spacing data from the deconvolved interferometer image is weighted down compared to the single dish image while the overlapping, longer spacing data from the single-dish are weighted down compared to the interferometer image.

Other Packages:

The `feather` task is analogous to the AIPS `IMERG` task and the MIRIAD `immerge` task with option `'feather'`.

The tapering uses the transform of the low resolution point spread function. This can be specified as an input image or the appropriate telescope beam for the single-dish. The point spread function for a single dish image may also be calculated using `clean`.

Advice: Note that if you are feathering large images, be advised to have the number of pixels along the X and Y axes to be composite numbers and definitely not prime numbers. In general FFTs

work much faster on even and composite numbers. You may use `subimage` function of the `image` tool to trim the number of pixels to something desirable.

The inputs for `feather` are:

```

imagenam  =      ''  # Name of output feathered image
highres   =      ''  # Name of high resolution (synthesis) image
lowres    =      ''  # Name of low resolution (single dish) image
async     =      False # If true the taskname must be started using feather(...)

```

Note that the only inputs are for images. Note that `feather` does not do any deconvolution but combines presumably deconvolved images after the fact.

Starting with a cleaned synthesis image and a low resolution image from a single dish telescope, the following example shows how they can be feathered:

```

feather(imagenam='feather.im',      # Create an image called feather.im
        highres='synth.im',         # The synthesis image is called synth.im
        lowres='single_dish.im'     # The SD image is called single_dish.im
        )

```

Note that the single dish image must have a well-defined beam shape and the correct flux units for a model image (Jy/beam instead of Jy/pixel) so use the tasks `imhead` and `immath` first to convert if needed.

5.5 Making Deconvolution Masks or Box Regions

For most careful imaging, you will want to restrict the region over which you allow CLEAN components to be found. To do this, you can create a 'deconvolution region' or 'mask' image using the `boxit` or `makemask` tasks. Note that `clean` can take simple boxes or box files as direct input to its `mask` parameter, so these tasks are most useful when direct input to `clean` (or use of interactive `clean`) will not suffice.

There are two ways to construct region files or mask images for use in deconvolution. The `boxit` task will find a set of box regions based upon an input image and control parameters. The `makemask` task will construct a mask image from a box specification or a box file.

5.5.1 Making Deconvolution Regions from an Image (`boxit`)

The `boxit` task creates "cleanbox" deconvolution regions automatically from an image. It searches the image to find "islands": all contiguous sets of pixels above the given threshold. The extreme x- and y-pixels of the island are used to determine the corners of a rectangular box that covers each island. The set of boxes are written out into a single region file with extension `.rgn`. `Boxit` works on single-plane images as well as multi-channel images: in the latter case, the thresholding and boxing is done separately in each plane of the image. The output region file from `boxit` can be used as the `mask` input parameter for the `clean` task (§ 5.3).

The parameter inputs for `boxit` are:

```
# boxit :: Box regions in image above given threshold value.
imagenam  =      '' # Name of image to threshold
regionfile =      '' # Output region file
threshold = '0.0mJy' # Threshold value. Must include units.
minsize   =      2 # Minimum number of pixels for a boxable island
diag      =      False # Count diagonal connections?
boxstretch =      1 # Increase box sizes by this many pixels beyond thresholded pixels.
overwrite =      False # Overwrite existing region file?
async     =      False # If true the taskname must be started using boxit(...)
```

The `regionfile` parameter specifies the root name of the region file. It will automatically be given `.rgn` as the file extension. The `minsize` parameter specifies the smallest island that qualifies to be boxed. It refers to the total number of pixels in the island. To include pixels connected only on the diagonal as being part of the same island, set the `diag` parameter to `True`. The `boxstretch` parameter increases the size of the boxes beyond the extent of the island, and can range from -1 to 5. For a value of 1 (the default), the box is stretched by one pixel in each outward direction; therefore, each side of the box lengthens by two pixels. Finally, the parameter `overwrite` specifies whether an existing region file can be overwritten.

ALERT: The `boxit` task is a prototype under active development and coded in Python. Eventually we will add functionality to deal with the creation of non-rectangular regions and with multi-plane masks, as well as efficiency improvements.

5.5.2 Making Deconvolution Masks from Boxes (`makemask`)

The parameter inputs for `makemask` are:

```
# makemask :: Derive a mask image from a cleanbox and set of imaging parameters:

cleanbox   =      [] # Clean box file or regions
vis        =      '' # Name of input visibility file (if no input image)
imagenam  =      '' # Name of output mask images
mode       =      'mfs' # Type of selection (mfs, channel, velocity)
imsize     = [256, 256] # Image size in spatial pixels [x,y]
cell       =      [1, 1] # Cell size in arcseconds
phasecenter =      '' # Field identifier or direction of the phase center
stokes     =      'I' # Stokes parameter to image (I,IV,IQU,IQUV)
field      =      '0' # Field ids list to use in mosaic
spw        =      '0' # Spectral window identifier (0-based)
```

The majority of the parameters are the standard imaging parameters (§ 5.2). The `cleanbox` parameter uses the syntax for cleanboxes as in the `clean` parameter `mask` (see § 5.3.6), with the option for `'interactive'` also. The `imagenam` parameter specifies the name for the output mask image.

You can use the `viewer` to figure out the `cleanbox` `blc-trc` x-y settings, make the mask image, and then bring it into the viewer as a contour image over your deconvolved image to compare exactly where your mask regions are relative to the actual emission. In this example, create a mask from many `cleanbox` regions specified in a file on disk (`cleanboxes.txt`) containing

```
1 80 80 120 120
2 20 40 24 38
3 70 42 75 66
```

where each line specifies the field index and the `blc` x-y and `trc` x-y positions of that `cleanbox`. For example, in `casapy`, you can do this easily:

```
CASA <29>: !cat > cleanboxes.txt
IPython system call: cat > cleanboxes.txt
1 80 80 120 120
2 20 40 24 38
3 70 42 75 66
<CNTRL-D>
CASA <30>: !cat cleanboxes.txt
IPython system call: cat cleanboxes.txt
1 80 80 120 120
2 20 40 24 38
3 70 42 75 66
```

Then, in `CASA`,

```
makemask(vis='source.ms',
         imagename='source.mask',
         cleanbox='cleanboxes.txt',
         mode='mfs',                # make a multi-frequency synthesis map (combine channels)
         insize=[200,200])         # Set image size = 200x200 pixels
         cell=[0.1,0.1],           # Using 0.1 arcsec pixels
         spw='0,1,2',              # Combine channels from 3 spectral windows
         field='0',                 # Use the first field in this split dataset
         stokes='I')               # Image stokes I polarization
```

This task will then create a mask image that has the 3 `cleanboxes` specified in the `cleanboxes.txt` file.

You can also specify the `cleanbox` as a list (of lists) of `blc, trc` pairs (4 vertices), e.g.

```
cleanbox = [[80, 80, 120, 120], [20, 40, 24, 38], [70, 42, 75, 66]]
```

is equivalent to the `cleanboxes.txt` given above. Likewise,

```
cleanbox = [80, 80, 120, 120]
```

puts in a single cleanbox.

Note that you must specify a visibility dataset and create the image properties so the mask image will have the same dimensions as the image you want to actually clean.

ALERT: Eventually we will add functionality to deal with the creation of non-rectangular regions and with multi-plane masks. There is also no `cleanbox='interactive'` version currently available. You have to run `clean` with `cleanbox='interactive'` to generate a mask graphically.

5.6 Transforming an Image Model (ft)

The `ft` task will Fourier transform an image and insert the resulting model into the `MODEL_DATA` column of a Measurement Set. You can also convert a CLEAN component list to a model and insert that into the `MODEL_DATA` column. The MS `MODEL_DATA` column is used, for example, to hold the model for calibration purposes in the tasks and toolkit. This is especially useful if you have a resolved calibrator and you want to start with a model of the source before you derive accurate gain solutions. This is also helpful for self-calibration (see § 5.8 below).

The inputs for `ft` are:

```
vis           =      '' # Name of input visibility file
fieldid       =      0 # Field index identifier
field         =      '' # Field name list
model         =      '' # Name of input model image
complist      =      '' # Name of component list
incremental   =      False # Add to the existing MODEL_DATA column?
```

An example of how to do this:

```
ft(vis='n75.ms',           # Start with the visibility dataset n75.ms
   field='1328',          # Select field name '1328+307' (minimum match)
   model='1328.model.image') # Name of the model image you have already
```

This task will Fourier transform the model image and insert the resulting model in the `MODEL_DATA` column of the rows of the MS corresponding to the source 1328+307.

Note that after `clean`, the transform of the final model is left in the `MODEL_DATA` column so you can go directly to a self-calibration step without explicitly using `ft`.

Inside the Toolkit:

The `im.ft` method does what the `ft` task does. Its main use is setting the `MODEL_DATA` column in the MS so that the `cb` tool can use it for subsequent calibration.

5.7 Image-plane deconvolution (deconvolve)

If you have only an image (obtained from some telescope) and an image of its point spread function, then you can attempt a simple image-plane deconvolution. Note that for interferometer data, full uv-plane deconvolution using `clean` or similar algorithm is superior!

The default inputs for `deconvolve` are:

```
# deconvolve :: Deconvolving a point spread function from an image

imagename =      '' # Name of image to deconvolve
model      =      '' # Name of output image to which deconvolved components are stored
psf        =      '' # Name of psf or gaussian parameters if psf is assumed gaussian
alg        =      'clark' # Deconvolution algorithm to use
niter      =      10 # number of iteration to use in deconvolution process
gain       =      0.1 # CLEAN gain parameter
threshold  =      '0.0Jy' # level below which sources will not be deconvolved
mask       =      '' # Name of image that has mask to limit region of deconvolution
async     =      False # if True run in the background, prompt is freed
```

The algorithm (`alg`) options are: `'clark'`, `'hogbom'`, `'multiscale'` or `'mem'`. The `'multiscale'` and `'mem'` options will open the usual set of sub-parameters for these methods.

5.8 Self-Calibration

Once you have a model image or set of model components reconstructed from your data using one of the deconvolution techniques described above, you can use it to refine your calibration. This is called *self-calibration* as it uses the data to determine its own calibration (rather than observations of special calibration sources).

In principle, self-calibration is no different than the calibration process we described earlier (§ 4). In effect, you alternate between calibration and imaging cycles, refining the calibration and the model as you go. The trick is you have to be careful, as defects in early stages of the calibration can get into the model, and thus prevent the calibration from improving. In practice, it is best to not clean very deeply early on, so that the CLEAN model contains correct components only.

One important thing to keep in mind is that the self-calibration relies upon having the most recent Fourier transform of the model in the `MODEL_DATA` column of the MS. This is indeed the case if you follow the imaging (using `clean`) directly by the self-calibration. If you have done something strange in between and have lost or overwritten the `MODEL_DATA` column (for example done some extra cleaning that you do not want to keep), then use the `ft` task (see § 5.6 above), which fills the `MODEL_DATA` column with the Fourier transform of the specified model or model image.

Likewise, during self-calibration (once you have a new calibration solution) the imaging part relies upon having the `CORRECTED_DATA` column contain the self-calibrated data. This is done with the `applycal` task (§ 4.6.1).

The `clearcal` command can be used during the self-calibration if you need to clear the `CORRECTED_DATA` column and revert to the original `DATA`. If you need to restore the `CORRECTED_DATA` to any previous stage in the self-calibration, use `applycal` again with the appropriate calibration tables.

ALERT: In later patches we will change the tasks so that users need not worry what is contained in the MS scratch columns and how to fill them. CASA will handle that underneath for you!

For now, we refer the user back to the calibration chapter for a reminder on how to run the calibration tasks.

See the example of cleaning and self-calibrating the Jupiter 6cm continuum data given below in Appendix F.2.

5.9 Automatic Cleaning

`Autoclean` is a task to automatically clean a dataset and create an output image by automating the “cleanbox” selection procedure. The cleaning and boxing progresses iteratively. The majority of the parameters are standard imaging parameters, with many added parameters that specify the boxing process and the stop conditions. Note that, like `clean` (§ 5.3), `autoclean` has the option to run interactively. In this case, the clean regions are still determined automatically, but they can be examined (and modified if necessary) in the `viewer` after each stage of boxing.

The parameter inputs for `autoclean` are:

```
# autoclean :: CLEAN an image with automatically-chosen clean regions.
vis                =      '' # name of input visibility file
imagename          =      '' # Pre-name of output images
field              =      '' # Field Name
spw                =      '' # Spectral windows:channels: '' is all
selectdata         =      False # Other data selection parameters
mode               =      'mfs' # Type of selection (mfs, channel, velocity, frequency)
psfmode            =      'clark' # method of PSF calculation to use during minor cycles
imagermode         =      '' # Use csclean or mosaic. If '', use psfmode
imsize             = [256, 256] # x and y image size in pixels, symmetric for single value
cell               = ['1.0arcsec', '1.0arcsec'] # x and y cell size. default unit arcsec
phasecenter        =      '' # Image phase center: position or field index
restfreq           =      '' # rest frequency to assign to image (see help)
stokes             =      'I' # Stokes params to image (currently requires Stokes="I")
weighting          =      'natural' # Weighting of uv (natural, uniform, briggs, ...)
interactive        =      False # use interactive clean (with GUI viewer)
mask               =      [] # Initial cleanbox(es), mask image(s), and/or region(s) used in clean
modelimage         =      '' # Name of model image(s) to initialize cleaning
uvtaper            =      False # Apply additional uv tapering of visibilities.
niter              =      500 # Max number of total clean cycles per channel
npercycle          =      100 # Clean iterations before adding new clean boxes
npercycle_speedup  =      1.5 # Increase npercycle by this factor if no new regions found.
gain               =      0.1 # Loop gain for cleaning
pbcor              =      False # Output primary beam-corrected image
minpb              =      0.1 # Minimum PB level to use
```

```

clean_threshold = '0.0mJy' # Flux level to stop cleaning. Must include units.
Nrms            =         6 # Threshold to stop cleaning: current rms * Nrms
eps_maxres     =         0.01 # Stop cleaning if max abs(residual) changes by less than this factor
useabsresid    =        False # If True, use absolute value of residual to test for clean convergen
allow_maxres_inc =         3 # Stop cleaning if max abs(residual) increases this many times (set t
island_rms     =         4 # Threshold for islands when finding clean regions: current rms * isl
diag           =        False # Count diagonal connection as same island?
peak_rms       =         6 # Threshold for island peak current rms * peak_rms
gain_threshold =         0.1 # Threshold for island peak current max residual * gain_threshold
Npeak          =         3 # Max number of new clean regions to add in each iteration.
shape          =         1 # Shape of clean regions: 0 for circles, 1 for boxes, 2 for both.
boxstretch     =         1 # Increase clean region size by this many pixels beyond island size.
irregsize      =        100 # For regions this large (in pixels), use irregular clean region.
async          =        False # If true the taskname must be started using autoclean(...)

```

Many of the parameters are the standard imaging parameters (§ 5.2). The new additions in general fall into three categories: clean cycle parameters, stopping parameters, and boxing parameters.

The new or modified clean cycle parameters are `niter`, `npercycle`, and `npercycle_speedup`. The `niter` parameter specifies the total number of clean minor cycles per image channel; `Autoclean` always cleans on a channel by channel basis. `Npercycle` specifies the initial number of minor clean cycles before the next stage of clean boxing. The `npercycle_speedup` parameter is a way to increase the value of `npercycle` if the cleaning is not deep enough. If no new clean regions are found, the value of `npercycle` increases by a factor of `npercycle_speedup`. Set this parameter to 1 to always keep the value of `npercycle` constant.

`Clean_threshold`, `Nrms`, `eps_maxres`, `useabsresid`, and `allow_maxres_inc` are parameters which determine the stopping conditions for cleaning in each image channel. Any single one of the following conditions is sufficient to discontinue cleaning the current image channel. The `clean_threshold` parameter is the flux level at which to stop cleaning. (It is equivalent to the `threshold` parameter for the `Clean` task.) The `Nrms` parameter is used to stop cleaning when the maximum residual is less than `Nrms` times the rms of the image. The maximum residual of the image is checked between each automated clean boxing cycle, and cleaning stops if the it has changed by a factor of less than `eps_maxres` from one cycle to the next. The maximum residual is allowed to increase between clean boxing cycles, but cleaning stops if it increases more than `allow_maxres_inc` times. And finally, the parameter `useabsresid`, primarily available for testing purposes, is set to `False` to use the (positive) maximum residual for these stopping conditions; set it to `True` to use the absolute value of the image residuals.

The parameters which determine the progression of the automated clean boxing are `island_rms`, `diag`, `peak_rms`, `gain_threshold`, `Npeak`, `shape`, `boxstretch`, and `irregsize`. The `diag` and `boxstretch` parameters are equivalent to those used in the `boxit` task (§ 5.5.1). `Island_rms` determines the threshold level for clean regions; the threshold (equivalent to `boxit` parameter `threshold`) is `island_rms` times the rms in the current residual image. A thresholded island only qualifies for a clean region if its largest pixel value is at least `peak_rms` times the image rms; this parameter should be larger than `island_rms`. The following two parameters are intended to avoid placing a clean region over a sidelobe of a bright source. `Npeak` strictly limits the number of new clean regions added in each automated boxing cycle, while the `gain_threshold` parameter ensures

that a thresholded island qualifies for a new clean region only if its maximum pixel is greater than `gain_threshold` times the current maximum residual.

The parameters `shape` and `irregsize` determine the shape of the clean regions that go into the mask. Set `shape` to 0 to make only circular regions, to 1 for only rectangular regions, and to 2 for either. If `shape = 2`, clean regions will be circular for islands whose extent in the x and y directions are within one pixel of each other, and will be rectangular for all others. Clean regions will have the exact (irregular) shape of the island for all islands which are larger than `irregsize` pixels in extent in either the x or y direction. To ignore this option, set `irregsize` to -1 (or to the image size). Note that `Autoclean` will produce a `.mask` image as well as a `.rgn` region file. The mask image will contain the full set of clean regions. The region file, owing to current CASA functionality, will contain only the rectangular regions. Eventually we will add functionality to deal with the creation of non-rectangular regions.

ALERT: The `autoclean` task is a prototype under active development and coded in Python. Eventually we will integrate this better with the clean task to improve efficiency. This task is provided for users who wish to experiment with this functionality, e.g. for pipeline development. We welcome feedback on the use of this tasks, in particular on heuristics used to set the control parameters.

5.10 Examples of Imaging

See the scripts provided in Appendix F for examples of imaging. In particular, we refer the interested user to the demonstrations for:

- NGC5921 (VLA HI) — a quick demo of basic CASA spectral line cube imaging and analysis (F.1)
- Jupiter (VLA 6cm continuum polarimetry) — polarization imaging and analysis (F.2)
- NGC4826 (BIMA 3mm CO) — mosaic imaging of spectral line data (F.3)

Chapter 6

Image Analysis

Once data has been calibrated (and imaged in the case of synthesis data), the resulting image or image cube must be displayed or analyzed in order to extract quantitative information, such as statistics or moment images. In addition, there need to be facilities for the coordinate conversion of images for direct comparison. We have assembled a skeleton of image analysis tasks for this release. Many more are still under development.

Inside the Toolkit:

Image analysis is handled in the `ia` tool. Many options exist there, including region statistics and image math. See § 6.12 below for more information.

The image analysis tasks are:

- `imhead` — summarize and manipulate the “header” information in a CASA image (§ 6.2)
- `imcontsub` — perform continuum subtraction on a spectral-line image cube (§ 6.3)
- `imfit` — image plane Gaussian component fitting (§ 6.4)
- `immath` — perform mathematical operations on or between images (§ 6.5)
- `immoments` — compute the moments of an image cube (§ 6.6)
- `imstat` — calculate statistics on an image or part of an image (§ 6.7)
- `imval` — extract the data and mask values from a pixel or region of an image (§ 6.8)
- `imregrid` — regrid an image onto the coordinate system of another image (§ 6.9)
- `imsmooth` — smooth images in the spectral and angular directions (§ 6.10)
- `importfits` — import a FITS image into a CASA *image* format table (§ 6.11.2)
- `exportfits` — write out an image in FITS format (§ 6.11.1)

There are other tasks which are useful during image analysis. These include:

- **viewer** — there are useful region statistics and image cube slice and profile capabilities in the viewer (§ 7)

We also give some examples of using the CASA Toolkit to aid in image analysis (§ 6.12).

6.1 Common Image Analysis Task Parameters

We now describe some sets of parameters are common to the image analysis. These should behave the same way in any of the tasks described in this section that they are found in.

6.1.1 Region Selection (box)

Area selection in the image analysis tasks is controlled by the **box** parameter or through the **regions** parameter (§ 6.1.5).

The **box** parameter selects rectangular areas:

```

box      =      '' # Select one or more box regions

#        string containing blcx,blcy,trcx,trcy

#        A box selection in the directional portion of an image.
#        The directional portion of an image are the axes for right
#        ascension and declination, for example. Boxes are specified
#        by there bottom-left corner (blc) and top-right corner (trc)
#        as follows: blcx, blcy, trcx, trcy;
#        ONLY pixel values acceptable at this time.
#        Default: none (all);
#        Example: box='0,0,50,50'
#        Example: box=' [10,20,30,40] ; [100,100,150,150] '
```

To get help on **box**, see the in-line help

```
help(par.box)
```

6.1.2 Plane Selection (chans, stokes)

The channel, frequency, or velocity plane(s) of the image is chosen using the **chans** parameter:

```

chans    =      '' # Select the channel(spectral) range

#        string containing channel range

#        immath, imstat, and imcontsub - takes a string listing
```

```
#           of channel numbers, velocity, and/or frequency
#           numbers, much like the spw paramter
#           Only channel numbers acceptable at this time.
#           Default: none (all);
#           Example: chans='3~20'
#                   chans="0,3,4,8"
#                   chans="3~20,50,51"
```

The polarization plane(s) of the image is chosen with the `stokes` parameter:

```
stokes      =      '' # Stokes params to image (I,IV,IQU,IQUV)

#           string containing Stokes selections

#           Stokes parameters to image, may or may not be separated
#           by commas but best if you use commas.
#           Default: none (all); Example: stokes='IQUV';
#           Example:stokes='I,Q'
#           Options: 'I','Q','U','V',
#                   'RR','RL','LR','LL',
#                   'XX','YX','XY','YY',...
```

To get help on these parameters, see the in-line help

```
help(par.chans)
help(par.stokes)
```

Sometimes, as in the `immoments` task, the channel/plane selection is generalized to work on more than one axis type. In this case, the `planes` parameter is used. This behaves like `chans` in syntax.

6.1.3 Lattice Expressions (`expr`)

Lattice expressions are strings that describe operations on a set of input images to form an output image. These strings use the *Lattice Expression Language* (LEL). LEL syntax is described in detail in AIPS++ Note 223

<http://aips2.nrao.edu/docs/notes/223/223.html>

ALERT: This document was written in the context of glish-based AIPS++ and is not yet updated to CASA syntax (see below).

The `expr` string contains the LEL expression:

```
expr        =      '' # Mathematical expression using images

#           string containing LEL expression
```

```

#           A mathematical expression, with image file names.
#           image file names must be enclosed in double quotes (")
#           Default: none
#           Example: expr='min("image2.im")+(2*max("image1.im"))'
#
#           Available functions in the expr and mask paramters:
#           pi(), e(), sin(), sinh(), asinh(), cos(), cosh(), tan(), tanh(),
#           atan(), exp(), log(), log10(), pow(), sqrt(), complex(), conj()
#           real(), imag(), abs(), arg(), phase(), aplitude(), min(), max()
#           round(), isgn(), floor(), ceil(), rebin(), spectralindex(), pa(),
#           iif(), indexin(), replace(), ...

```

For examples using LEL `expr`, see § 6.5.1 below. Note that in `immath`, shortcut names have been given to the images provided by the user in `imagename` that can be used in the LEL expression, for the above example:

```

imagename=['image2.im','image1.im']
expr='min(IM0)+(2*max(IM1))'

```

ALERT: As of Patch 2, LEL expressions use 0-based indices. Also, the functions must be lowercase (in almost all cases we know about).

6.1.4 Masks (`mask`)

The `mask` string contains a LEL expression (see § 6.1.3 above). This string can be an on-the-fly (OTF) mask expression or refer to an image pixel mask.

```

mask           =           '' # Mask to be applied to the images

#           string containing LEL expression

#           Name of mask applied to each image in the calculation
#           Default '' means no mask;
#           Example: mask='ngc5921.clean.cleanbox.mask">0.5'
#           mask='mask(ngc5921.clean.cleanbox.mask)'

```

Note that the mask file supplied in the `mask` parameter must have the same shape, same number of axes and same axes length, as the images supplied in the `expr` parameter, with one exception. The mask may be missing some of the axes — if this is the case then the mask will be expanded along these axes to become the same shape.

For examples using `mask`, see § 6.5.2 below.

6.1.5 Regions (`region`)

The `region` parameter points to an ImageRegion file. An ImageRegion file can be created with the CASA `viewer`'s region manager (§ 7.3.5). Typically ImageRegion files will have the suffix `'.rgn'`. If a region file is given then the `box`, `chans`, and `stokes` parameters will be ignored.

For example:

```
region='myimage.im.rgn'
```

6.2 Image Header Manipulation (imhead)

To summarize and change keywords and values in the “header” of your image, use the `imhead` task. Its inputs are:

```
# imhead :: Lists, gets and puts image header parameters
imagename    =      ''    # Name of input image file
mode         =  'summary' # Options: get, put, summary, list, stats
async       =      False
```

The `mode` parameter controls the operation of `imhead`.

Setting `mode='summary'` will print out a summary of the image properties and the header to the logger.

Setting `mode='list'` prints out a list of the header keywords and values to the terminal.

The `mode='get'` allows the user to retrieve the current value for a specified keyword `hdkey`:

```
mode         =      'get'  # imhead options: list, summary, get, put
hdkey        =      ''    # The FITS keyword
```

Note that to catch this value, you need to assign it to a Python variable. See § 1.3.3 for more on return values.

The `mode='put'` allows the user to replace the current value for a given keyword `hditem` with that specified in `hdvalue`. There are two sub-parameters that are opened by this option:

```
mode         =      'put'  # imhead options: list, summary, get, put
hdkey        =      ''    # The FITS keyword
hdvalue      =      ''    # Value of hdkey
hdtype       =      ''    # Data type of the header keyword.
hdcomment    =      ''    # Comment associated with the header keyword
```

WARNING: Be careful when using `mode='put'`. This task does no checking on whether the values you specify (e.g. for the axes types) are valid, and you can render your image invalid. Make sure you know what you are doing when using this option!

6.2.1 Examples for imhead

The following uses the example images from NGC5921 (§ F.1).

We can print the summary to the logger:

```
CASA <51>: imhead('ngc5921.demo.cleaning.image',mode='summary')
```

prints in the logger:

```
##### Begin Task: imhead          #####
Image name       : ngc5921.demo.cleaning.image
Object name      : N5921_2
Image type       : PagedImage
Image quantity   : Intensity
Pixel mask(s)    : None
Region(s)        : None
Image units      : Jy/beam
Restoring Beam   : 52.3782 arcsec, 45.7319 arcsec, -165.572 deg

Direction reference : J2000
Spectral reference  : LSRK
Velocity type      : RADIO
Rest frequency     : 1.42041e+09 Hz
Pointing center    : 15:22:00.000000 +05.04.00.000000
Telescope          : VLA
Observer           : TEST
Date observation   : 1995/04/13/00:00:00
Telescope position: [-1.60119e+06m, -5.04198e+06m, 3.55488e+06m] (ITRF)

Axis Coord Type   Name                Proj Shape Tile  Coord value at pixel  Coord incr Units
-----
0  0  Direction Right Ascension  SIN   256   64  15:22:00.000   128.00 -1.500000e+01 arcsec
1  0  Direction Declination      SIN   256   64 +05.04.00.000   128.00  1.500000e+01 arcsec
2  1  Stokes Stokes              1     1                I
3  2  Spectral Frequency         46    8   1.41279e+09    0.00  2.4414062e+04 Hz
      Velocity                 1607.99  0.00 -5.152860e+00 km/s

##### End Task: imhead
```

If you choose `mode='list'`, you get the summary in the logger and a listing of keywords and values to the terminal:

```
CASA <52>: imhead('ngc5921.demo.cleaning.image',mode='list')
Out [52]:
{'beammajor': 52.378242492675781,
 'beamminor': 45.731891632080078,
 'beampa': -165.5721435546875,
 'bunit': 'Jy/beam',
 'cdelt1': '-7.27220521664e-05',
 'cdelt2': '7.27220521664e-05',
```

```

'cdelt3': '1.0',
'cdelt4': '24414.0625',
'crpix1': 128.0,
'crpix2': 128.0,
'crpix3': 0.0,
'crpix4': 0.0,
'crval1': '4.02298392585',
'crval2': '0.0884300154344',
'crval3': 'I',
'crval4': '1412787144.08',
'ctype1': 'Right Ascension',
'ctype2': 'Declination',
'ctype3': 'Stokes',
'ctype4': 'Frequency',
'cunit1': 'rad',
'cunit2': 'rad',
'cunit3': '',
'cunit4': 'Hz',
'datamax': ' Not Known ',
'datamin': -0.010392956435680389,
'date-obs': '1995/04/13/00:00:00',
'equinox': 'J2000',
'imtype': 'Intensity',
'masks': ' Not Known ',
'maxpixpos': array([134, 134,  0,  38], dtype=int32),
'maxpos': '15:21:53.976, +05.05.29.998, I, 1.41371e+09Hz',
'minpixpos': array([117,  0,  0,  21], dtype=int32),
'minpos': '15:22:11.035, +04.31.59.966, I, 1.4133e+09Hz',
'object': 'N5921_2',
'observer': 'TEST',
'projection': 'SIN',
'reffreqtype': 'LSRK',
'restfreq': [1420405752.0],
'telescope': 'VLA'}

```

Note that this list is a return value and can be captured in a variable:

```
mylist = imhead('ngc5921.demo.cleanimg.image',mode='list')
```

The values for these keywords can be queried using `mode='get'`. At this point you should capture the return value:

```
CASA <53>: mybmaj = imhead('ngc5921.demo.cleanimg.image',mode='get',hdkey='beammajor')
```

```
CASA <54>: mybmaj
Out[54]: {'unit': 'arcsec', 'value': 52.378242492699997}
```

```
CASA <55>: myobserver = imhead('ngc5921.demo.cleanimg.image',mode='get',hdkey='observer')
```

```
CASA <56>: print myobserver
{'value': 'TEST', 'unit': ''}
```

You can set the values for these keywords using `mode='put'`. For example:

```
CASA <57>: imhead('ngc5921.demo.cleanimg.image',mode='put',hdkey='observer',hdvalue='CASA')
Out[57]: 'CASA'
```

```
CASA <58>: imhead('ngc5921.demo.cleanimg.image',mode='get',hdkey='observer')
Out[58]: {'unit': '', 'value': 'CASA'}
```

6.3 Continuum Subtraction on an Image Cube (`imcontsub`)

One method to separate line and continuum emission in an image cube is to specify a number of line-free channels in that cube, make a linear fit to the visibilities in those channels, and subtract the fit from the whole cube. Note that the task `uvcontsub` serves a similar purpose; see § 4.7.4 for a synopsis of the pros and cons of either method.

The `imcontsub` task will subtract a polynomial baseline fit to the specified channels from an image cube.

The default inputs are:

```
# imcontsub :: Continuum subtraction on images
imasename = '' # Name of the input image
linefile = '' # Output line image file name
contfile = '' # Output continuum image file name
fitorder = 0 # Polynomial order for the continuum estimation
region = '' # Image region or name to process see viewer
box = '' # Select one or more box regions
chans = '' # Select the channel(spectral) range
stokes = '' # Stokes params to image (I,IV,IQU,IQUV)
async = False
```

Area selection using `box` and `region` is detailed in § 6.1.1 and § 6.1.5 respectively.

Image cube plane selection using `chans` and `stokes` are described in § 6.1.2.

ALERT: `imcontsub` has issues when the image does not contain a spectral or stokes axis. Errors are generated when run on an image missing one or both of these axes. You will need to use the Toolkit (e.g. the `ia.adddegaxes` method) to add degenerate missing axes to the image.

6.3.1 Examples for `imcontsub`)

The following uses the example images from NGC5921 (§ F.1).

First, we make a clean image without the uv-plane continuum subtraction:

```
# First, run clearcal to clear the uvcontsub results from the
# corrected column
```

```

clearcal('ngc5921.demo.src.split.ms')

# Now clean, keeping all the channels except first and last
default('clean')
vis = 'ngc5921.demo.src.split.ms'
imagenname = 'ngc5921.demo.nouvcontsub'
mode = 'channel'
nchan = 61
start = 1
width = 1
imsize = [256,256]
psfmode = 'clark'
imagermode = ''
cell = [15.,15.]
niter = 6000
threshold='8.0mJy'
weighting = 'briggs'
robust = 0.5
mask = [108,108,148,148]
interactive=False
clean()

# It will have made the image:
# -----
# ngc5921.demo.nouvcontsub.image

# You can view this image
viewer('ngc5921.demo.nouvcontsub.image')

```

You can clearly see continuum sources in the image which were removed previously in the script by the use of `uvcontsub`. Lets see if `imcontsub` can work as well.

Using the viewer, it looks like channels 0 through 4 and 50 through 60 are line-free. Then:

```

default('imcontsub')
imagenname = 'ngc5921.demo.nouvcontsub.image'
linefile = 'ngc5921.demo.nouvcontsub.lineimage'
contfile = 'ngc5921.demo.nouvcontsub.contimage'
fitorder = 1
chans = '0~4,50~60'
stokes = 'I'
imcontsub()

```

This did not do too badly!

6.4 Image-plane Component Fitting (`imfit`)

The inputs are:


```

#                                     'value': 0.00046695916589535109}},
#                                     'm0': {'unit': 'rad', 'value': -2.0541102061078207},
#                                     'm1': {'unit': 'rad', 'value': 1.1439131060384089},
#                                     'refer': 'J2000',
#                                     'type': 'direction'},
#   'majoraxis': {'unit': 'arcsec', 'value': 0.29100166137741568},
#   'majoraxiserror': {'unit': 'arcsec',
#                       'value': 0.0011186420613222663},
#   'minoraxis': {'unit': 'arcsec', 'value': 0.24738110059830495},
#   'minoraxiserror': {'unit': 'arcsec',
#                       'value': 0.0013431999725066338},
#   'positionangle': {'unit': 'deg', 'value': 19.369249322401796},
#   'positionangleerror': {'unit': 'rad',
#                           'value': 0.016663189295782171},
#   'type': 'Gaussian'},
# 'spectrum': {'frequency': {'m0': {'unit': 'GHz', 'value': 1.0},
#                              'refer': 'LSRK',
#                              'type': 'frequency'},
#              'type': 'Constant'}}

# Now the other components
xfit_B_res = imfit('b1608.demo.clean2.image', box='108,114,120,126',
                  newestimates='b1608.demo.clean2.newestimate', append=True)
xfit_B = xfit_B_res['results']['component0']

xfit_C_res = imfit('b1608.demo.clean2.image', box='108,84,120,96')
xfit_C = xfit_C_res['results']['component0']

xfit_D_res = imfit('b1608.demo.clean2.image', box='144,98,157,110')
xfit_D = xfit_D_res['results']['component0']

print ""
print "Imfit Results:"
print "-----"
print "A Flux = %6.4f Bmaj = %6.4f" % (xfit_A['flux']['value'][0], xfit_A['shape']['majoraxis']['value'])
print "B Flux = %6.4f Bmaj = %6.4f" % (xfit_B['flux']['value'][0], xfit_B['shape']['majoraxis']['value'])
print "C Flux = %6.4f Bmaj = %6.4f" % (xfit_C['flux']['value'][0], xfit_C['shape']['majoraxis']['value'])
print "D Flux = %6.4f Bmaj = %6.4f" % (xfit_D['flux']['value'][0], xfit_D['shape']['majoraxis']['value'])
print ""

```

Now try fitting four components together. For this we will have to provide an estimate file. We will use the clean beam for the estimate of the component sizes:

```

estfile=open('b1608.demo.clean2.estimate','w')
print >>estfile,'# peak, x, y, bmaj, bmin, bpa'
print >>estfile,'0.017, 128, 129, 0.293arcsec, 0.238arcsec, 21.7deg'
print >>estfile,'0.008, 113, 120, 0.293arcsec, 0.238arcsec, 21.7deg'
print >>estfile,'0.008, 113, 90, 0.293arcsec, 0.238arcsec, 21.7deg'
print >>estfile,'0.002, 151, 104, 0.293arcsec, 0.238arcsec, 21.7deg'
estfile.close()

```

Then, this can be used in `imfit`:

```
xfit_all_res = imfit('b1608.demo.clean2.image',
                    estimates='b1608.demo.clean2.estimate',
                    logfile='b1608.demo.clean2.imfitall.log',
                    newestimates='b1608.demo.clean2.newestimate',
                    box='121,121,136,136,108,114,120,126,108,84,120,96,144,98,157,110')
# Now extract the fit part of the return values
xfit_allA = xfit_all_res['results']['component0']
xfit_allB = xfit_all_res['results']['component1']
xfit_allC = xfit_all_res['results']['component2']
xfit_allD = xfit_all_res['results']['component3']
```

These results are almost identical to those from the individual fits. You can see a nicer printout of the fit results in the logfile.

6.5 Mathematical Operations on an Image (`immath`)

The inputs are:

```
# immath :: Perform math operations on images
imagename      =      ''      # a list of input images
mode           = 'evalexpr'   # mode for math operation (evalexpr, spix, pola, poli)
  expr         =      ''      # Mathematical expression using images
  varnames     =      ''      # a list of variable names to use with the image files

outfile        = 'immath_results.im' # File where the output is saved
mask           =      ''      # Mask to be applied to the images
region         =      ''      # File path which contains an Image Region
box            =      ''      # Select one or more box regions in the input images
chans          =      ''      # Select the channel(spectral) range
stokes         =      ''      # Stokes params to image (I,IV,IQU,IQUV)
async         =      False    # If true the taskname must be started using immath(...)
```

In all cases, `outfile` must be supplied with the name of the new output file to create.

The `mode` parameter selects what `immath` is to do.

The default `mode='evalexpr'` lets the user specify a mathematical operation to carry out on one or more input images. The sub-parameter `expr` contains the Lattice Expression Language (LEL) string describing the image operations based on the images in the `imagename` parameter. See § 6.1.3 for more on LEL strings and the `expr` parameter.

Mask specification is done using the `mask` parameter. This can optionally contain an on-the-fly mask expression (in LEL) or point to an image with a pixel mask. See § 6.1.4 for more on the use of the `mask` parameter. See also § 6.1.3 for more on LEL strings.

Region selection is carried out through the `region` and `box` parameters. See § 6.1.1 and § 6.1.5 for more on area selection.

Image plane selection is controlled by `chans` and `stokes`. See § 6.1.2 for details on plane selection.

For `mode='evalexpr'`, the standard usage for specifying images to be used in the LEL expression is to provide them as a list in the `imagename` parameter, and then access them in the LEL expression by the names `IM0`, `IM1`, For example,

```
immath(imagename=['image1.im','image2.im'],expr='IM0-IM1',outfile='ImageDiff.im')
```

would subtract the second image given from the first.

For the special modes `'spix'`, `'pola'`, `'poli'`, the required images for the given operation are to be provided in `imagename` (some times in a particular order). **V3.0 ALERT:** For `mode='pola'` you MUST call as a function as in the example below (§ 6.5.1.2), giving the parameters as arguments, or `immath` will fail.

Detailed examples are given below.

6.5.1 Examples for `immath`

The following are examples using `immath` using `NGC5921` (§ F.1). Note that the image names in the `expr` are assumed to refer to existing image files in the current working directory.

6.5.1.1 Simple math

Select a single plane (channel 22) of the 3-D cube and subtract it from the original image:

```
immath(imagename='ngc5921.demo.cleanimg.image',
      expr='IM0',chans='22',
      outfile='ngc5921.demo.chan22.image')
```

Double all values in our image:

```
immath(imagename=['ngc5921.demo.chan22.image'],
      expr='IM0*2.0',
      outfile='ngc5921.demo.chan22double.image' )
```

Square all values in our image:

```
immath(imagename=['ngc5921.demo.chan22.image'],
      expr='IM0^2',
      outfile='ngc5921.demo.chan22squared.image' )
```

Note that the units in the output image are still claimed to be “Jy/beam”, ie. `immath` will not correctly scale the units in the image for non-linear cases like this. Beware.

You can do other mathematical operations on an image (e.g. trigonometric functions) as well as use scalar results from an image (e.g. max, min, median, mean, variance). You also have access to constants such as `e()` and `pi()` (which are doubles internally, while most images are floats). For example: Take the sine of an image:

```
immath(imagename=['ngc5921.demo.chan22.image', 'ngc5921.demo.chan22squared.image'],
       expr='sin(float(pi())/IM0/sqrt(max(IM1)))',
       outfile='ngc5921.demo.chan22sine.image')
```

Note again that the units are again kept as they were.

Select a single plane (channel 22) of the 3-D cube and subtract it from the original image:

```
immath(imagename='ngc5921.demo.cleanimg.image',
       expr='IM0', chans='22',
       outfile='ngc5921.demo.chan22.image')
```

```
immath(imagename=['ngc5921.demo.cleanimg.image', 'ngc5921.demo.chan22.image'],
       expr='IM0-IM1',
       outfile='ngc5921.demo.sub22.image')
```

Note that in this example the 2-D plane gets expanded out and the values are applied to each plane in the 3-D cube.

Select and save the inner 1/4 of an image for channels 40,42,44 as well as channels 10 and below:

```
default('immath')
imagename=['ngc5921.demo.cleanimg.image']
expr='IM0'
box='64,64,192,192'
chans='<10;40,42,44'
outfile='ngc5921.demo.inner.image'
immath()
```

ALERT: Note that if `chan` selects more than one channel then the output image has a number of channels given by the span from the lowest and highest channel selected in `chan`. In the example above, it will have 45 channels. The ones not selected will be masked in the output cube. If we had set

```
chans = '40,42,44'
```

then there would be 5 output channels corresponding to channels 40,41,42,43,44 of the MS with 41,43 masked. Also, the `chans='<10'` selects channels 0–9.

Note that the `chans` syntax allows the operators '<', '<=', '>', '>'. For example,

```
chans = '<17,>79'
chans = '<=16,>=80'
```

do the same thing.

Divide an image by another, with a threshold on one of the images:

```
immath(imagename=['ngc5921.demo.cleanimg.image', 'ngc5921.demo.chan22.image'],
       expr='IM0/IM1[IM1>0.008]',
       outfile='ngc5921.demo.div22.image')
```

6.5.1.2 Polarization manipulation

The following are some examples using the 3C129 Tutorial

```
http://casa.nrao.edu/Doc/Scripts/3c129_tutorial.py
```

as an example.

It is helpful to extract the Stokes planes from the cube into individual images:

```
default('immath')
imagename = '3C129BC.clean.image'
outfile='3C129BC.I'; expr='IM0'; stokes='I'; immath();
outfile='3C129BC.Q'; expr='IM0'; stokes='Q'; immath();
outfile='3C129BC.U'; expr='IM0'; stokes='U'; immath();
outfile='3C129BC.V'; expr='IM0'; stokes='V'; immath();
```

Extract linearly polarized intensity and polarization position angle images:

```
immath(stokes='', outfile='3C129BC.P', mode='poli',
       imagename=['3C129BC.Q', '3C129BC.U'], sigma='0.0mJy/beam');
immath(stokes='', outfile='3C129BC.X', mode='pola',
       imagename=['3C129BC.Q', '3C129BC.U'], sigma='0.0mJy/beam');
```

V3.0 ALERT: For mode='pola' you MUST call as a function as in this example (giving the parameters as arguments) or `immath` will fail.

Create a fractional linear polarization image:

```
default('immath')
imagename = ['3C129BC.I', '3C129BC.Q', '3C129BC.U']
outfile='3C129BC.fractional_linpol'
expr='sqrt((IM1^2 + IM2^2)/IM0^2)'
stokes=''
immath()
```

Create a polarized intensity image:

```
default('immath')
imagename = ['3C129BC.Q', '3C129BC.U', '3C129BC.V']
outfile='3C129BC.pol_intensity'
expr='sqrt(IM0^2 + IM1^2 + IM2^2)'
stokes=''
immath()
```

Toolkit Tricks: The following uses the toolkit (§ 6.12). You can make a complex linear polarization ($Q + iU$) image using the `imagepol` tool:

```

# See CASA User Reference Manual:
# http://casa.nrao.edu/docs/casaref/imagepol-Tool.html
#
# Make an imagepol tool and open the clean image
potool = casac.homefinder.find_home_by_name('imagepolHome')
po = potool.create()
po.open('3C129BC.clean.image')
# Use complexlinpol to make a Q+iU image
po.complexlinpol('3C129BC.cmplxlinpol')
po.close()

```

You can now display this in the viewer, in particular overlay this over the intensity raster with the intensity contours. When you load the image, use the LEL:

```
'3C129BC.cmplxlinpol' ['3C129BC.P'>0.0001]
```

which is entered into the LEL box at the bottom of the Load Data menu (§ 7.2.4).

6.5.1.3 Primary beam correction/uncorrection

In a script using `mode='evalexpr'`, you might want to assemble the string for `expr` using string variables that contain the names of files. Since you need to include quotes inside the `expr` string, use a different quote outside (or escape the string, e.g. `'5`. For example, to do a primary beam correction on the NGC5921 cube,

```

imname = 'ngc5921.demo.cleanimg'
imagenname = imname
...
clean()

default('immath')
clnimage = imname + '.image'
fluximage = imname + '.flux'
pbcorimage = imname + '.pbcor'

outfile = pbcorimage
imagenname = [clnimage,fluximage]
# cutoff at the 10% level
expr='IM0/IM1[IM1>0.1]'

immath()

```

Note that we did not use a `minpb` when we cleaned, so we use the trick above to effectively set a cutoff in the primary beam `.flux` image of 0.1.

For more on LEL strings, see AIPS++ Note 223

<http://aips2.nrao.edu/docs/notes/223/223.html>

or in § 6.1.3 above.

6.5.1.4 Spectral analysis

One can make an integrated 1-d spectrum over the whole image by rebinning (integrating) over the two coordinate axes in a specified region. For example, using the NGC5921 image cube (with 46 channels):

```
immath(outfile="ngc5921.demo.spectrum.all",mode="evalexpr",
       imagename='ngc5921.demo.clean.image',
       expr="rebin(IM0,[256,256,1,1])")
```

The resulting image has shape `[1,1,1,46]` as desired. You can view this with the `viewer` and will see a 1-D spectrum.

One can also do this with a box:

```
immath(outfile="ngc5921.demo.spectrum.box",mode="evalexpr",
       imagename='ngc5921.demo.clean.image',
       expr="rebin(IM0,[256,256,1,1])",box="118,118,141,141")
```

ALERT: One cannot specify a `region` without it collapsing the channel axis (even when told to use all axes or channels).

Toolkit Tricks: The following uses the toolkit (§ 6.12). You can make an ascii file containing only the values (no other info though):

```
ia.open('ngc5921.demo.spectrum.all')
ia.toASCII('ngc5921.demo.spectrum.all.ascii')
```

You can also extract to a record inside Python:

```
myspec = ia.torecord()
```

which you can then manipulate in Python.

6.5.2 Using masks in `immath`

The `mask` parameter is used inside `immath` to apply a mask to all the images used in `expr` before calculations are done (if you are curious, it uses the `ia.subimage` tool method to make virtual images that are then input in the LEL to the `ia.imagecalc` method).

For example, lets assume that we have made a single channel image using `clean` for the NGC5921 data (see Appendix F.1).

```
default('clean')

vis = 'ngc5921.demo.src.split.ms.contsub'
imagenam = 'ngc5921.demo.chan22.cleaning'
```



```

mode = 'channel'
nchan = 1
start = 22
step = 1

field = ''
spw = ''
imsize = [256,256]
cell = [15.,15.]
psfalg = 'clark'
gain = 0.1
niter = 6000
threshold='8.0mJy'
weighting = 'briggs'
rmode = 'norm'
robust = 0.5

mask = [108,108,148,148]

clean()

```

There is now a file 'ngc5921.demo.chan22.cleanimg.mask' that is an image with values 1.0 inside the cleanbox region and 0.0 outside.

We can use this to mask the clean image:

```

default('immath')
imagenname = 'ngc5921.demo.chan22.cleanimg.image'
expr='IMO'
mask='ngc5921.demo.chan22.cleanimg.mask">0.5'
outfile='ngc5921.demo.chan22.cleanimg.imasked'
immath()

```

Toolbox Tricks: Note that there are also *pixel masks* that can be contained in each image. These are Boolean masks, and are implicitly used in the calculation for each image in `expr`. If you want to use the mask in a different image not in `expr`, try it in `mask`:

```

# First make a pixel mask inside ngc5921.demo.chan22.cleanimg.mask
ia.open('ngc5921.demo.chan22.cleanimg.mask')
ia.calcmask('ngc5921.demo.chan22.cleanimg.mask">0.5')
ia.summary()
ia.close()
# There is now a 'mask0' mask in this image as reported by the summary

# Now apply this pixel mask in immath
default('immath')
imagenname='ngc5921.demo.chan22.cleanimg.image'
expr='IMO'
mask='mask(ngc5921.demo.chan22.cleanimg.mask)'
outfile='ngc5921.demo.chan22.cleanimg.imasked1'
immath()

```

Note that nominally the axes of the mask must be congruent to the axes of the images in `expr`. However, one exception is that the image in `mask` can have *fewer* axes (but not axes that exist but are of the wrong lengths). In this case `immath` will extend the missing axes to cover the range in the images in `expr`. Thus, you can apply a mask made from a single channel to a whole cube.

```
# drop degenerate stokes and freq axes from mask image
ia.open('ngc5921.demo.chan22.cleanimg.mask')
im2 = ia.subimage(outfile='ngc5921.demo.chan22.cleanimg.mymask', dropdeg=True)
im2.summary()
im2.close()
ia.close()
# mymask has only RA and Dec axes

# Now apply this mask to the whole cube
default('immath')
imagename='ngc5921.demo.cleanimg.image'
expr='IMO'
mask='"ngc5921.demo.chan22.cleanimg.mymask">0.5'
outfile='ngc5921.demo.cleanimg.imasked'
immath()
```

For more on masks as used in LEL, see

<http://aips2.nrao.edu/docs/notes/223/223.html>

or in § 6.1.4 above.

6.6 Computing the Moments of an Image Cube (`immoments`)

For spectral line datasets, the output of the imaging process is an `image cube`, with a frequency or velocity channel axis in addition to the two sky coordinate axes. This can be most easily thought of as a series of `image planes` stacked along the spectral dimension.

A useful product to compute is to collapse the cube into a *moment* image by taking a linear combination of the individual planes:

$$M_m(x_i, y_i) = \sum_k^N w_m(x_i, y_i, v_k) I(x_i, y_i, v_k) \quad (6.1)$$

for pixel i and channel k in the cube I . There are a number of choices to form the m moment, usually approximating some polynomial expansion of the intensity distribution over velocity mean or sum, gradient, dispersion, skew, kurtosis, etc.). There are other possibilities (other than a weighted sum) for calculating the image, such as median filtering, finding minima or maxima along the spectral axis, or absolute mean deviations. And the axis along which to do these calculation need not be the spectral axis (ie. do moments along Dec for a RA-Velocity image). We will treat all of these as generalized instances of a “moment” map.

The `immoments` task will compute basic moment images from a cube. The default inputs are:

```

# immoments :: Compute moments of an image cube:
imagename  =      '' # Input image name
moments    =      [0] # List of moments you would like to compute
axis       = 'spectral' # The moment axis: ra, dec, lat, long, spectral, or stokes
region     =      '' # Image Region. Use viewer
box        =      '' # Select one or more box regions
chans      =      '' # Select the channel(spectral) range
stokes     =      '' # Stokes params to image (I,IV,IQU,IQV)
mask       =      '' # mask used for selecting the area of the image to calculate the moments on
includepix =      -1 # Range of pixel values to include
excludepix =      -1 # Range of pixel values to exclude
outfile    =      '' # Output image file name (or root for multiple moments)
async      =      False # If true the taskname must be started using immoments(...)

```

This task will operate on the input file given by `imagename` and produce a new image or set of images based on the name given in `outfile`.

The `moments` parameter chooses which moments are calculated. The choices for the operation mode are:

```

moments=-1 - mean value of the spectrum
moments=0  - integrated value of the spectrum
moments=1  - intensity weighted coordinate;traditionally used to get
             'velocity fields'
moments=2  - intensity weighted dispersion of the coordinate; traditionally
             used to get 'velocity dispersion'
moments=3  - median of I
moments=4  - median coordinate
moments=5  - standard deviation about the mean of the spectrum
moments=6  - root mean square of the spectrum
moments=7  - absolute mean deviation of the spectrum
moments=8  - maximum value of the spectrum
moments=9  - coordinate of the maximum value of the spectrum
moments=10 - minimum value of the spectrum
moments=11 - coordinate of the minimum value of the spectrum

```

The meaning of these is described in the CASA Reference Manual:

<http://casa.nrao.edu/docs/casaref/image.moments.html>

The `axis` parameter sets the axis along which the moment is “collapsed” or calculated. Choices are: `'ra'`, `'dec'`, `'lat'`, `'long'`, `'spectral'`, or `'stokes'`. A standard moment-0 or moment-1 image of a spectral cube would use the default choice `'spectral'`. One could make a position-velocity map by setting `'ra'` or `'dec'`.

The `includepix` and `excludepix` parameters are used to set ranges for the inclusion and exclusion of pixels based on values. For example, `includepix=[0.05,100.0]` will include pixels with values from 50 mJy to 1000 Jy, and `excludepix=[100.0,1000.0]` will exclude pixels with values from 100 to 1000 Jy.

If a single moment is chosen, the `outfile` specifies the exact name of the output image. If multiple `moments` are chosen, then `outfile` will be used as the root of the output filenames, which will get different suffixes for each moment.

6.6.1 Hints for using (`immoments`)

In order to make an unbiased moment-0 image, do not put in any thresholding using `includepix` or `excludepix`. This is so that the (presumably) zero-mean noise fluctuations in off-line parts of the image cube will cancel out. If your image has large biases, like a pronounced clean bowl due to missing large-scale flux, then your moment-0 image will be biased also. It will be difficult to alleviate this with a threshold, but you can try.

To make a usable moment-1 (or higher) image, on the other hand, it is critical to set a reasonable threshold to exclude noise from being added to the moment maps. Something like a few times the rms noise level in the usable planes seems to work (put into `includepix` or `excludepix` as needed. Also use `chans` to ignore channels with bad data.

6.6.2 Examples using (`immoments`)

For example, using the NGC5921 example (§ F.1):

```
default('immoments')
imasename = 'ngc5921.demo.cleaning'
# Do first and second spectral moments
axis = 'spectral'
chans = ''
moments = [0,1]
# Need to mask out noisy pixels, currently done
# using hard global limits
excludepix = [-100,0.009]
outfile = 'ngc5921.demo.moments'

immoments()

# It will have made the images:
# -----
# ngc5921.demo.moments.integrated
# ngc5921.demo.moments.weighted_coord
```

Other examples of NGC2403 (a moment zero image of a VLA line dataset) and NGC4826 (a moment one image of a BIMA CO line dataset) are shown in Figure 6.1.

ALERT: We are working on improving the thresholding of planes beyond the global cutoffs in `includepix` and `excludepix`.

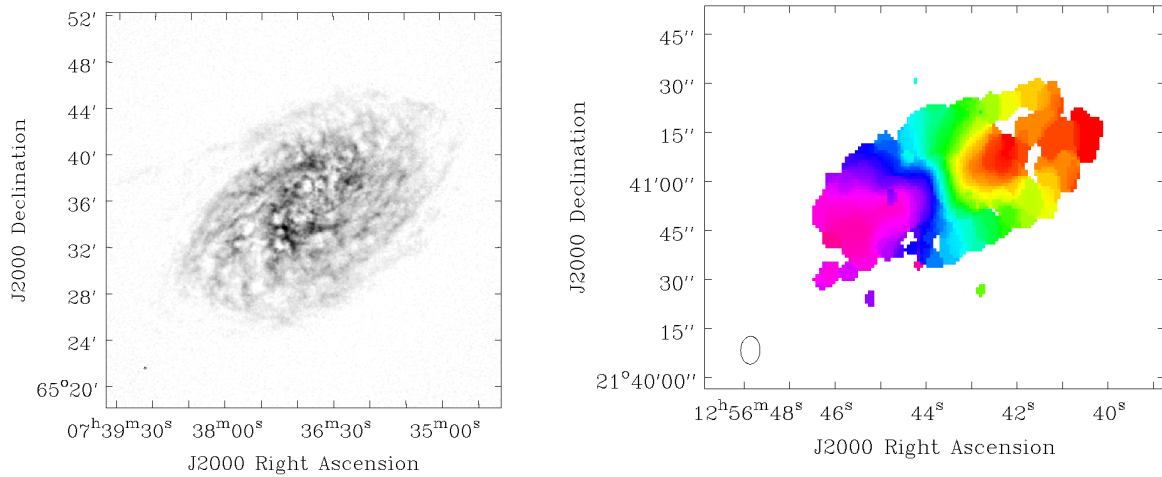


Figure 6.1: NGC2403 VLA moment zero (left) and NGC4826 BIMA moment one (right) images as shown in the viewer.

6.7 Computing image statistics (imstat)

The `imstat` task will calculate statistics on a region of an image, and return the results as a return value in a Python dictionary.

The inputs are:

```
# imstat :: Displays statistical information on an image or image region
imasename = '' # Name of the input image
region = '' # Image Region or name. Use Viewer
box = '' # Select one or more box regions
chans = '' # Select the channel(spectral) range
stokes = '' # Stokes params to image (I,IV,IQU,IQV)
async = False
```

Area selection using `box` and `region` is detailed in § 6.1.1 and § 6.1.5 respectively.

Plane selection is controlled by `chans` and `stokes`. See § 6.1.2 for details on plane selection.

ALERT: As with `imcontsub`, if the image is missing one or more of the stokes and spectral axes, then `imstat` will fail. See the discussion of the workaround for this in § 6.3.

6.7.1 Using the task return value

The contents of the return value of `imstat` are in a Python dictionary of key-value sets. For example,

```
xstat = imstat()
```

will assign this to the Python variable `xstat`.

The keys for `xstat` are then:

```
KEYS
blc      - absolute PIXEL coordinate of the bottom left corner of
          the bounding box surrounding the selected region
blcf     - Same as blc, but uses WORLD coordinates instead of pixels
trc      - the absolute PIXEL coordinate of the top right corner
          of the bounding box surrounding the selected region
trcf     - Same as trc, but uses WORLD coordinates instead of pixels
flux     - the integrated flux density if the beam is defined and
          the if brightness units are $Jy/beam$
npts     - the number of unmasked points used
max      - the maximum pixel value
min      - minimum pixel value
maxpos   - absolute PIXEL coordinate of maximum pixel value
maxposf  - Same as maxpos, but uses WORLD coordinates instead of pixels
minpos   - absolute pixel coordinate of minimum pixel value
minposf  - Same as minpos, but uses WORLD coordinates instead of pixels
sum      - the sum of the pixel values:  $\sum I_i$ 
sumsq    - the sum of the squares of the pixel values:  $\sum I_i^2$ 
mean     - the mean of pixel values:
           $\bar{I} = \sum I_i / n$ 
sigma    - the standard deviation about the mean:
           $\sigma^2 = (\sum I_i - \bar{I})^2 / (n-1)$ 
rms      - the root mean square:
           $\sqrt{\sum I_i^2 / n}$ 
median   - the median pixel value (if robust=T)
medabsdevmed - the median of the absolute deviations from the
          median (if robust=T)
quartile - the inter-quartile range (if robust=T). Find the points
          which are 25% largest and 75% largest (the median is
          50% largest), find their difference and divide that
          difference by 2.
```

For example, an `imstat` call might be

```
default('imstat')
imagename = 'ngc5921.demo.cleanimg.image' # The NGC5921 image cube
box       = '108,108,148,148'           # 20 pixels around the center
chans     = '21'                       # channel 21

xstat = imstat()
```

In the terminal window, `imstat` reports:

Statistics on ngc5921.usecase.clean.image

Region ---

```
-- bottom-left corner (pixel) [blc]: [108, 108, 0, 21]
-- top-right corner (pixel) [trc]: [148, 148, 0, 21]
-- bottom-left corner (world) [blcf]: 15:22:20.076, +04.58.59.981, I, 1.41332e+09Hz
-- top-right corner( world) [trcf]: 15:21:39.919, +05.08.59.981, I, 1.41332e+09Hz
```

Values --

```
-- flux [flux]: 0.111799236126
-- number of points [npts]: 1681.0
-- maximum value [max]: 0.029451508075
-- minimum value [min]: -0.00612453464419
-- position of max value (pixel) [maxpos]: [124, 131, 0, 21]
-- position of min value (pixel) [minpos]: [142, 110, 0, 21]
-- position of max value (world) [maxposf]: 15:22:04.016, +05.04.44.999, I, 1.41332e+09Hz
-- position of min value (world) [minposf]: 15:21:45.947, +04.59.29.990, I, 1.41332e+09Hz
-- Sum of pixel values [sum]: 1.32267159822
-- Sum of squared pixel values [sumsq]: 0.0284534543692
```

Statistics ---

```
-- Mean of the pixel values [mean]: 0.000786836167885
-- Standard deviation of the Mean [sigma]: 0.00403944306904
-- Root mean square [rms]: 0.00411418313161
-- Median of the pixel values [median]: 0.000137259965413
-- Median of the deviations [medabsdevmed]: 0.00152346317191
-- Quartile [quartile]: 0.00305395200849
```

The return value in xstat is

CASA <152>: xstat

```
Out[152]:
{'blc': array([108, 108, 0, 21]),
 'blcf': '15:22:20.076, +04.58.59.981, I, 1.41332e+09Hz',
 'flux': array([ 0.11179924]),
 'max': array([ 0.02945151]),
 'maxpos': array([124, 131, 0, 21]),
 'maxposf': '15:22:04.016, +05.04.44.999, I, 1.41332e+09Hz',
 'mean': array([ 0.00078684]),
 'medabsdevmed': array([ 0.00152346]),
 'median': array([ 0.00013726]),
 'min': array([-0.00612453]),
 'minpos': array([142, 110, 0, 21]),
 'minposf': '15:21:45.947, +04.59.29.990, I, 1.41332e+09Hz',
 'npts': array([ 1681.]),
 'quartile': array([ 0.00305395]),
 'rms': array([ 0.00411418]),
 'sigma': array([ 0.00403944]),
 'sum': array([ 1.3226716]),
 'sumsq': array([ 0.02845345]),
```

```
'trc': array([148, 148,  0, 21]),
'trcf': '15:21:39.919, +05.08.59.981, I, 1.41332e+09Hz'}
```

ALERT: The return dictionary currently includes NumPy `array` values, which have to be accessed by an array index to get the array value. To access these dictionary elements, use the standard Python dictionary syntax, e.g.

```
xstat[<key string>][<array index>]
```

For example, to extract the standard deviation as a number

```
mystddev = xstat['sigma'][0]
print 'Sigma = '+str(xstat['sigma'][0])
```

6.7.2 Examples for `imstat`)

The following are some examples using the B1608+656 Tutorial

```
http://casa.nrao.edu/Doc/Scripts/b1608\_demo.py
```

as an example.

To extract statistics for the final image:

```
xstat = imstat('b1608.demo.clean2.image')
# Printing out some of these
print 'Max   = '+str(xstat['max'][0])
print 'Sigma = '+str(xstat['sigma'][0])
# results:
# Max   = 0.016796965152
# Sigma = 0.00033631979385
```

In a box around the brightest component:

```
xstat_A = imstat('b1608.demo.clean2.image', box='124,125,132,133')
# Printing out some of these
print 'Comp A Max Flux = '+str(xstat_A['max'][0])
print 'Comp A Max X,Y = ('+str(xstat_A['maxpos'][0])+', '+str(xstat_A['maxpos'][1])+')'
# results:
# Comp A Max Flux = 0.016796965152
# Comp A Max X,Y = (128,129)
```


6.8 Extracting data from an image (imval)

The `imval` task will extract the values of the data and mask from a specified region of an image and place in the task return value as a Python dictionary.

The inputs are:

```
# imval :: Get the data value(s) and/or mask value in an image.
imasename = '' # Name of the input image
region = '' # Image Region. Use viewer
box = '' # Select one or more box regions
chans = '' # Select the channel(spectral) range
stokes = '' # Stokes params to image (I,IV,IQU,IQUV)
async = False
```

Area selection using `box` and `region` is detailed in § 6.1.1 and § 6.1.5 respectively. By default, `box=''` will extract the image information at the reference pixel on the direction axes.

Plane selection is controlled by `chans` and `stokes`. See § 6.1.2 for details on plane selection. By default, `chans=''` and `stokes=''` will extract the image information in all channels and Stokes planes.

For instance,

```
xval = imval('myimage', box='144,144', stokes='I' )
```

will extract the Stokes I value or spectrum at pixel 144,144, while

```
xval = imval('myimage', box='134,134.154,154', stokes='I' )
```

will extract a 21 by 21 pixel region.

Extractions are returned in NumPy arrays in the return value dictionary, plus some extra elements describing the axes and selection:

```
CASA <2>: xval = imval('ngc5921.demo.moments.integrated')
```

```
CASA <3>: xval
```

```
Out[3]:
{'axes': [[0, 'Right Ascension'],
          [1, 'Declination'],
          [3, 'Frequency'],
          [2, 'Stokes']],
 'blc': [128, 128, 0, 0],
 'data': array([ 0.89667124]),
 'mask': array([ True], dtype=bool),
 'trc': [128, 128, 0, 0],
 'unit': 'Jy/beam.km/s'}
```

extracts the reference pixel value in this 1-plane image. Note that the 'data' and 'mask' elements are NumPy arrays, not Python lists.

To extract a spectrum from a cube:

```
CASA <8>: xval = imval('ngc5921.demo.clean.image',box='125,125')
```

```
CASA <9>: xval
```

```
Out[9]:
{'axes': [[0, 'Right Ascension'],
          [1, 'Declination'],
          [3, 'Frequency'],
          [2, 'Stokes']],
 'blc': [125, 125, 0, 0],
 'data': array([ 8.45717848e-04,  1.93370355e-03,  1.53750915e-03,
                2.88399984e-03,  2.38683447e-03,  2.89159478e-04,
                3.16268904e-03,  9.93389636e-03,  1.88773088e-02,
                3.01138610e-02,  3.14478502e-02,  4.03211266e-02,
                3.82498614e-02,  3.06552909e-02,  2.80734301e-02,
                1.72479432e-02,  1.20884273e-02,  6.13593217e-03,
                9.04005766e-03,  1.71429547e-03,  5.22095338e-03,
                2.49114982e-03,  5.30831399e-04,  4.80734324e-03,
                1.19265869e-05,  1.29435991e-03,  3.75700940e-04,
                2.34788167e-03,  2.72604497e-03,  1.78467855e-03,
                9.74952069e-04,  2.24676146e-03,  1.82263291e-04,
                1.98463408e-06,  2.02975096e-03,  9.65532148e-04,
                1.68218743e-03,  2.92119570e-03,  1.29359076e-03,
                -5.11484570e-04,  1.54162932e-03,  4.68662125e-04,
                -8.50282842e-04, -7.91683051e-05,  2.95954203e-04,
                -1.30133145e-03]),
 'mask': array([ True,  True,  True,  True,  True,  True,  True,  True,  True,  True,
                True,  True,  True,  True,  True,  True,  True,  True,  True,  True,
                True,  True,  True,  True,  True,  True,  True,  True,  True,
                True,  True,  True,  True,  True,  True,  True,  True,  True,  True], dtype=bool),
 'trc': [125, 125, 0, 45],
 'unit': 'Jy/beam'}
```

To extract a region from the plane of a cube:

```
CASA <13>: xval = imval('ngc5921.demo.clean.image',box='126,128,130,129',chans='23')
```

```
CASA <14>: xval
```

```
Out[14]:
{'axes': [[0, 'Right Ascension'],
          [1, 'Declination'],
          [3, 'Frequency'],
          [2, 'Stokes']],
 'blc': [126, 128, 0, 23],
 'data': array([[ 0.00938627,  0.01487772],
                [ 0.00955847,  0.01688832],
```

```

        [ 0.00696965,  0.01501907],
        [ 0.00460964,  0.01220793],
        [ 0.00358087,  0.00990202]]),
'mask': array([[ True,  True],
               [ True,  True],
               [ True,  True],
               [ True,  True],
               [ True,  True]], dtype=bool),
'trc': [130, 129, 0, 23],
'unit': 'Jy/beam'}

```

```

CASA <15>: print xval['data'][0][1]
0.0148777160794

```

In this example, a rectangular box was extracted, and you can see the order in the array and how to address specific elements.

6.9 Regridding an Image (imregrid)

It is occasionally necessary to regrid an image onto a new coordinate system. The `imregrid` task will regrid one image onto the coordinate system of another, creating an output image. In this task, the user need only specify the names of the input, template, and output images.

If the user needs to do more complex operations, such as regridding an image onto an arbitrary (but known) coordinate system, changing from Equatorial to Galactic coordinates, or precessing Equinoxes, the CASA toolkit can be used (see sidebar). Some of these facilities will eventually be provided in task form.

The default inputs are:

```

# imregrid :: regrid an image onto a template image
imasename      =          ''          # Name of input image
template       =          ''          # Name of reference image
output         =          ''          # Name of output regridded image
async          =          False       #

```

The output image will have the data in `imasename` regridded onto the coordinate system of `template` image.

ALERT: The `imregrid` task is currently under test against similar AIPS tasks and we are looking to improve its performance. Future releases will enable regridding without the need for a template image.

Inside the Toolkit:

More complex coordinate system and image regridding operation can be carried out in the toolkit. The `coordsys (cs)` tool and the `ia.regrid` method are the relevant components.

6.10 Image Convolution(`imsmooth`)

The default inputs are:

```
# imsmooth :: Smooth an image or portion of an image
imasename = '' # Name of the input image
kernel = 'boxcar' # Type of kernel to use: gaussian or boxcar.
region = '' # Image Region or name. Use viewer
box = '' # Select one or more box regions
chans = '' # Select the channel(spectral) range
stokes = '' # Stokes params to image (I,IV,IQU,IQV)
mask = '' # mask used for selecting the area of the image to smooth
outfile = '' # Output, smoothed, image file name
async = False #
```

6.11 Image Import/Export to FITS

These tasks will allow you to write your CASA image to a FITS file that other packages can read, and to import existing FITS files into CASA as an image.

6.11.1 FITS Image Export (`exportfits`)

To export your images to fits format use the `exportfits` task. The inputs are:

```
# exportfits :: Convert a CASA image to a FITS file
imasename = '' # Name of input CASA image
fitsimage = '' # Name of output image FITS file
velocity = False # Use velocity (rather than frequency) as spectral axis
optical = False # Use the optical (rather than radio) velocity convention
bitpix = -32 # Bits per pixel
minpix = 0 # Minimum pixel value
maxpix = 0 # Maximum pixel value
overwrite = False # Overwrite pre-existing imasename
dropstokes = False # Drop the Stokes axis?
stokeslast = True # Put Stokes axis last in header?
async = False # If true the taskname must be started using exportfits(...)
```

The `dropstokes` or `stokeslast` parameter may be needed to make the FITS image compatible with an external application.

For example,

```
exportfits('ngc5921.demo.cleanimg.image', 'ngc5921.demo.cleanimg.image.fits')
```

6.11.2 FITS Image Import (importfits)

You can also use the `importfits` task to import a FITS image into CASA image table format. Note, the CASA viewer can read fits images so you don't need to do this if you just want to look at the image. The inputs for `importfits` are:

```
# importfits :: Convert an image FITS file into a CASA image:

fitsimage =      '' # Name of input image FITS file
imagename  =      '' # Name of output CASA image
whichrep   =      0 # Which coordinate representation (if multiple)
whichhdu   =      0 # Which image (if multiple)
zeroblanks =      True # If blanked fill with zeros (not NaNs)
overwrite  =      False # Overwrite pre-existing imagename
async     =      False # if True run in the background, prompt is freed
```

For example, we can read the above image back in

```
importfits('ngc5921.demo.cleanimg.image.fits','ngc5921.demo.cleanimage')
```

6.12 Using the CASA Toolkit for Image Analysis

Although this cookbook is aimed at general users employing the tasks, we include here a more detailed description of doing image analysis in the CASA toolkit. This is because there are currently only a few tasks geared towards image analysis, as well as due to the breadth of possible manipulations that the toolkit allows that more sophisticated users will appreciate.

To see a list of the `ia` methods available, use the CASA `help` command:

```
CASA <1>: help ia
-----> help(ia)
Help on image object:

class image(__builtin__.object)
| image object
|
| Methods defined here:
|
| __init__(...)
|     x.__init__(...) initializes x; see x.__class__.__doc__ for signature
|
| __str__(...)
|     x.__str__() <==> str(x)
```

Inside the Toolkit:

The image analysis tool (`ia`) is the workhorse here. It appears in the User Reference Manual as the `image` tool. Other relevant tools for analysis and manipulation include `measures` (`me`), `quanta` (`qa`) and `coordsys` (`cs`).

```

|
|  adddegaxes(...)
|      Add degenerate axes of the specified type to the image'  :
|      outfile
|      direction = false
|      spectral  = false
|      stokes
|      linear    = false
|      tabular   = false
|      overwrite = false
|      -----
|
|  addnoise(...)
|
...
|
|  unlock(...)
|      Release any lock on the image'  :
|      -----
|
|  -----
|  Data and other attributes defined here:
|
|  __new__ = <built-in method __new__ of type object at 0x55d0f20>
|      T.__new__(S, ...) -> a new object with type S, a subtype of T

```

or for a compact listing use <TAB> completion on `ia.`, e.g.

```

CASA <2>: ia.
Display all 101 possibilities? (y or n)
ia.__class__          ia.fitsky             ia.newimagefromshape
ia.__delattr__       ia.fromarray         ia.open
ia.__doc__           ia.fromascii        ia.outputvariant
ia.__getattr__       ia.fromfits         ia.pixelvalue
ia.__hash__         ia.fromforeign      ia.putchunk
ia.__init__         ia.fromimage        ia.putregion
ia.__new__          ia.fromshape        ia.rebin
ia.__reduce__       ia.getchunk         ia.regrid
ia.__reduce_ex__    ia.getregion        ia.remove
ia.__repr__         ia.getslice         ia.removefile
ia.__setattr__      ia.hanning          ia.rename
ia.__str__          ia.haslock          ia.replacemaskedpixels
ia.adddegaxes       ia.histograms       ia.restoringbeam
ia.addnoise         ia.history           ia.rotate
ia.boundingBox      ia.imagecalc        ia.sepconvolve
ia.brightnessunit   ia.imageconcat      ia.set
ia.calc             ia.insert           ia.setboxregion
ia.calcmask         ia.isopen           ia.setbrightnessunit
ia.close            ia.ispersistent     ia.setcoordsys

```

ia.continuumsb	ia.lock	ia.sethistory
ia.convertflux	ia.makearray	ia.setmiscinfo
ia.convolve	ia.makecomplex	ia.setrestoringbeam
ia.convolve2d	ia.maketestimage	ia.shape
ia.coordmeasures	ia.maskhandler	ia.statistics
ia.coordsys	ia.maxfit	ia.subimage
ia.decompose	ia.miscinfo	ia.summary
ia.deconvolvecomponentlist	ia.modify	ia.toASCII
ia.done	ia.moments	ia.tofits
ia.echo	ia.name	ia.topixel
ia.fft	ia.newimage	ia.toworld
ia.findsources	ia.newimagefromarray	ia.twopointcorrelation
ia.fitallprofiles	ia.newimagefromfile	ia.type
ia.fitpolynomial	ia.newimagefromfits	ia.unlock
ia.fitprofile	ia.newimagefromimage	

A common use of the `ia` tool is to do region statistics on an image. The `imhead` task has `mode='stats'` to do this quickly over the entire image cube. The tool can do this on specific planes or sub-regions. For example, in the Jupiter 6cm example script (§ F.2), the `ia` tool is used to get on-source and off-source statistics for regression:

```
# The variable clnimage points to the clean image name

# Pull the max and rms from the clean image
ia.open(clnimage)
on_statistics=ia.statistics()
thistest_immax=on_statistics['max'][0]
oldtest_immax = 1.07732224464
print ' Clean image ON-SRC max should be ',oldtest_immax
print ' Found : Max in image = ',thistest_immax
diff_immax = abs((oldtest_immax-thistest_immax)/oldtest_immax)
print ' Difference (fractional) = ',diff_immax

print ''
# Now do stats in the lower right corner of the image
box = ia.setboxregion([0.75,0.00],[1.00,0.25],frac=true)
off_statistics=ia.statistics(region=box)
thistest_imrms=off_statistics['rms'][0]
oldtest_imrms = 0.0010449
print ' Clean image OFF-SRC rms should be ',oldtest_imrms
print ' Found : rms in image = ',thistest_imrms
diff_imrms = abs((oldtest_imrms-thistest_imrms)/oldtest_imrms)
print ' Difference (fractional) = ',diff_imrms

print ''
print ' Final Clean image Dynamic Range = ',thistest_immax/thistest_imrms
print ''
print ' ===== '

ia.close()
```

ALERT: Bad things can happen if you open some tools, like `ia`, in the Python command line on files and forget to close them before running scripts that use the `os.system('rm -rf <filename>')` call to clean up. We are in the process of cleaning up cases like this where there can be stale handles on files that have been manually deleted, but for the meantime be warned that you might get exceptions (usually of the “`SimpleOrderedMap-remove`” flavor, or even Segmentation Faults and core-dumps!

6.13 Examples of CASA Image Analysis

See the scripts provided in Appendix F for examples of data and image analysis. In particular, we refer the interested user to the demonstrations for:

- NGC5921 (VLA HI) — a quick demo of basic CASA spectral line analysis (F.1)
- Jupiter (VLA 6cm continuum polarimetry) — polarization image analysis (F.2)

Chapter 7

Visualization With The CASA Viewer

This chapter describes how to display data with the `casaviewer` either as a stand-alone or through the `viewer` task. You can display both images and Measurement Sets.

7.1 Starting the viewer

Within the `casapy` environment, the `viewer` task can be used to display an image or MS. The inputs are:

```
# viewer :: View an image or visibility data set.

infile      =          '' # (Optional) Name of file to visualize.
displaytype = 'raster'   # (Optional) Type of visual rendering
                                     # (raster, contour, vector or marker).
                                     # lel if an lel expression is given
                                     # for infile (advanced).
```

Examples of starting the viewer:

```
CASA <1>: viewer()

CASA <2>: viewer('ngc5921.usecase.ms')

CASA <3>: viewer('ngc5921.usecase.clean.image')

CASA <4>: viewer('ngc5921.usecase.clean.image.rstr')

CASA <5>: viewer('ngc5921.usecase.clean.image', 'contour')

CASA <6>: viewer('"ngc5921.usecase.clean.image"^2', 'lel')
```

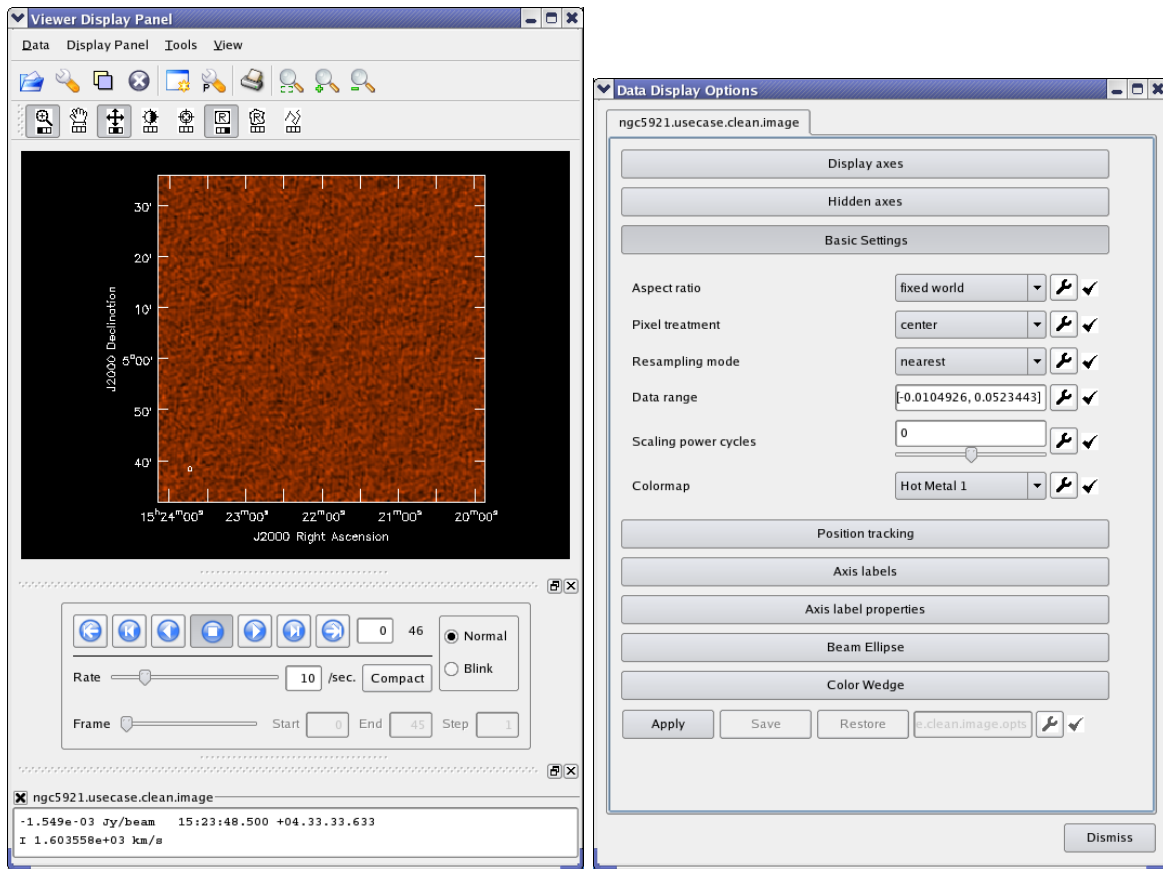


Figure 7.1: The **Viewer Display Panel** (left) and **Data Display Options** (right) panels that appear when the viewer is called with the image cube from NGC5921 (`viewer('ngc5921.usecase.clean.image')`). The initial display is of the first channel of the cube.

The first of these creates an empty **Viewer Display Panel** (§ 7.2.1) and a **Load Data** window (§ 7.2.4). The second starts the `viewer` loaded with a Measurement Set. The third example starts the `viewer` with an image cube (see Figure 7.1).

Example four brings up a display panel as it was when its state was saved to the given 'restore' file (`ngc5921.usecase.clean.image.rstr`). This includes the data displayed as well as options and viewer settings. (See § 7.2.2, **Saving and Restoring Viewer State**).

Examples five and six are less common cases, which make use of the second parameter (`displaytype`). Example five displays the image in contour form. Example six uses 'Lattice (Image) Expression Language' to display the square of the image data.

NOTE: the `viewer` task now determines file types (images, MSs, restore files) automatically. It is no longer necessary to specify `filetype='ms'` explicitly.

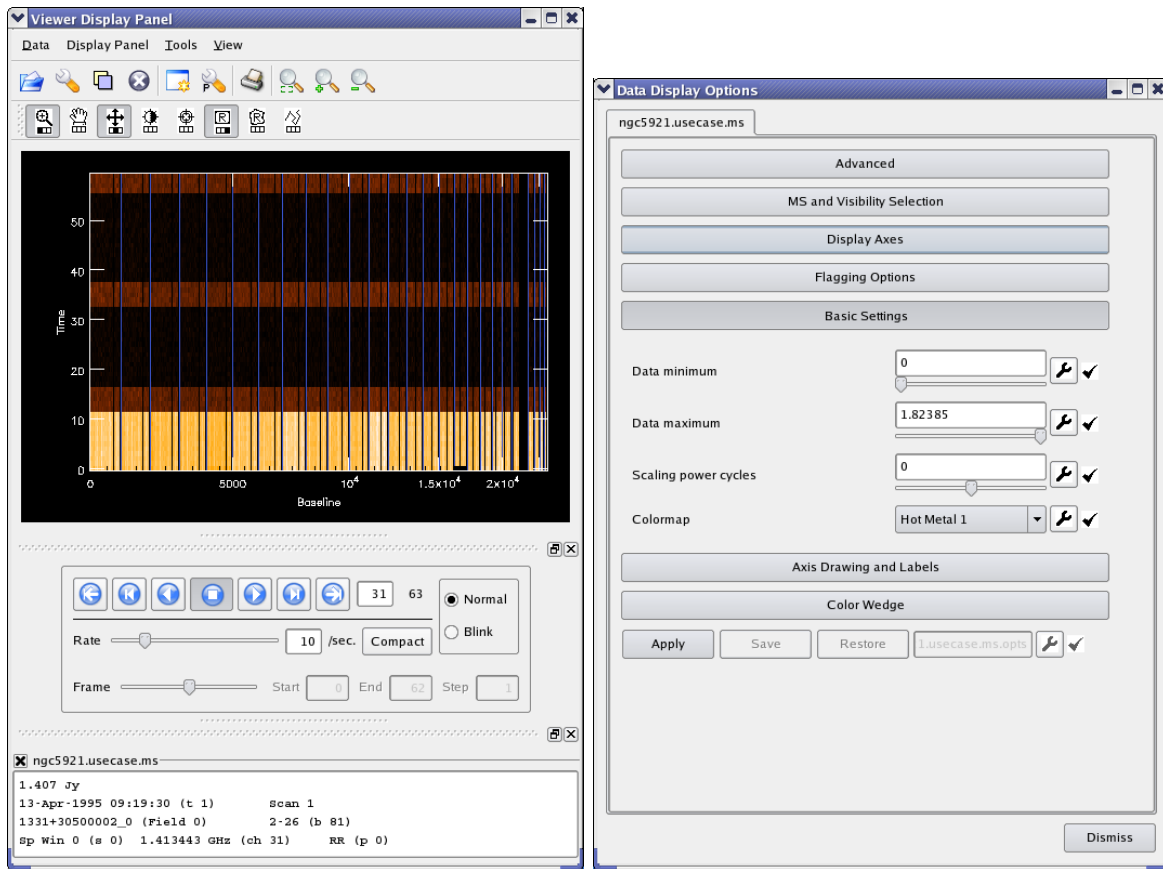


Figure 7.2: The **Viewer Display Panel** (left) and **Data Display Options** (right) panels that appear when the viewer is called with the NGC5921 Measurement Set (`viewer('ngc5921.usecase.ms')`).

7.1.1 Running the CASA viewer outside casapy

`casaviewer` is the name of the stand-alone viewer application that is available with a CASA installation. From the operating system prompt, the following commands are equivalent to the `casapy` task commands given previously:

```
casaviewer &
casaviewer ms_filename &
casaviewer image_filename &
casaviewer restore_filename &
```

```
casaviewer image_filename contour &
casaviewer '"image_filename"^2' l1l1 &
```

7.2 The viewer GUI

The CASA **viewer** application consists of a number of graphical user interface (GUI) windows that respond to mouse and keyboard input. Here we describe the **Viewer Display Panel** (§ 7.2.1) and the **Load Data** window (§ 7.2.4). They are used for both image and MS viewing. Several other windows are context-specific and are described in the sections on viewing images (§ 7.3) and Measurement Sets (§ 7.4).

7.2.1 The Viewer Display Panel

The Viewer Display Panel is the the window that actually displays the image or MS. This is shown in the left panels of Figures 7.1 and 7.2. Note that this panel is the same whether an image or MS is being displayed.

At the top of the Viewer Display Panel are the menus:

- **Data**
 - **Open** — choose a data file to load and display
 - **Register** — select/de-select the (previously-loaded) data file(s) which should display right now (menu expands to the right showing all loaded data)
 - **Close** — close (unload) selected data file (menu expands to the right)
 - **Adjust** — open the Data Display Options ('Adjust') panel
 - **Print** — print the displayed image
 - **Save Panel State** — to a 'restore' file (xml format)
 - **Restore Panel State** — from a restore file
 - **Close Panel** — close the Viewer Display Panel (will exit if this is the last display panel open)
 - **Quit Viewer** — close all display panels and exit
- **Display Panel**
 - **New Panel** — create another Viewer Display Panel (cleared)
 - **Panel Options** — open the Display Panel's options window
 - **Save Panel State**
 - **Restore Panel State**
 - **Print** — print displayed image

- **Close Panel** — close the Viewer Display Panel (will exit if this is the last display panel open)
- **Tools**
 - **Box in File** — save/load and set parameters of boxes that are stored in external files
 - **Region in File** — save/load and set parameters of regions that are stored in external files
 - **Region in Image** — save/load and set parameters of boxes that are stored in the image itself
 - **Spectral Profile** — plot frequency/velocity profile of point or region of image
 - **Shape Manager** — save regions and control their extent
- **View**
 - **Main Toolbar** — show/hide top row of icons
 - **Mouse Toolbar** — show/hide second row of mouse-button action selection icons
 - **Animator** — show/hide tapedeck control panel
 - **Position Tracking** — show/hide bottom position tracking report box

Below this is the **Main Toolbar** (Figure 7.3), the top row of icons for fast access to some of these menu items:

- **folder** (Data:Open shortcut) — show the Load Data panel
- **wrench** (Data:Adjust shortcut) — show the Data Display Options ('Adjust') panel
- **panels** (Data:Register shortcut) — show the menu of loaded data
- **delete** (Data:Close shortcut) — closes/unloads selected data
- **new panel** (Display Panel:New Panel)
- **panel wrench** (Display Panel:Panel Options) — show the Display Panel's options window
- **save** — save panel state to a 'restore' file
- **restore** — restore panel state from a restore file regions.
(Note: some of these newer buttons do not appear in older figures of this document).
- **print** (Display Panel:Print) — print data
- **magnifier box** — Zoom out all the way
- **magnifier plus** — Zoom in (by a factor of 2)
- **magnifier minus** — Zoom out (by a factor of 2)



Figure 7.3: The display panel's **Main Toolbar** appears directly below the menus and contains 'shortcut' buttons for most of the frequently-used menu items.



Figure 7.4: The '**Mouse Tool**' Bar allows you to assign separate mouse buttons to tools you control with the mouse within the image display area. Initially, zooming, color adjustment, and rectangular regions are assigned to the left, middle and right mouse buttons, respectively.

Below this are the eight **Mouse Tool** buttons (Figure 7.4). These allow assignment of *each* of the three mouse buttons to a different operation on the display area. Clicking a mouse tool icon will [re-]assign **the mouse button that was clicked** to that tool. The icons show which mouse button is currently assigned to which tool.

The 'escape' key can be used to cancel any mouse tool operation that was begun but not completed, and to erase any tool showing in the display area.

- **Zooming (magnifying glass icon):** To zoom into a selected area, press the Zoom tool's mouse button (the **left** button by default) on one corner of the desired rectangle and drag to the desired opposite corner. Once the button is released, the zoom rectangle can still be moved or resized by dragging. To complete the zoom, double-click inside the selected rectangle (double-clicking *outside* it will zoom *out* instead).
- **Panning (hand icon):** Press the tool's mouse button on a point you wish to move, drag it to the position where you want it moved, and release. *Note: The arrow keys, Page Up, Page Down, Home and End keys can also be used to scroll through your data any time you are zoomed in. (Click on the main display area first, to be sure the keyboard is 'focused' there).*
- **Stretch-shift colormap fiddling (crossed arrows):** This is usually the handiest color adjustment; it is assigned to the **middle** mouse button by default.
- **Brightness-contrast colormap fiddling (light/dark sun)**
- **Positioning (bombsight):** This tool can place a 'crosshair' marker on the display to select a position. It is used to flag Measurement Set data or to select an image position for spectral profiles. Click on the desired position with the tool's mouse button to place the crosshair; once placed you can drag it to other locations. Double-click is not needed for this tool. See § 7.2.3 for more detail.

- **Rectangle and Polygon region drawing:** The rectangle region tool is assigned to the **right** mouse button by default. As with the zoom tool, a rectangle region is generated by dragging with the assigned mouse button; the selection is confirmed by double-clicking within the rectangle. Polygon regions are created by clicking the assigned mouse button at the desired vertices, clicking the final location twice to finish. Once created, a polygon can be moved by dragging from inside, or reshaped by dragging the handles at the vertices. Double-click inside to confirm region selection. See § 7.2.3 for the uses of this tool.
- **Polyline drawing:** A polyline can be created by selecting this tool. It is manipulated similarly to the polygon region tool: create segments by clicking at the desired positions and then double-click to finish the line. [Uses for this tool are still to be implemented].

The main **Display Area** lies below the toolbars.

Underneath the display area is an **Animator** panel. The most prominent feature is the “tape deck” which provides movement between image planes along a selected third dimension of an image cube. This set of buttons is only enabled when a registered image reports that it has more than one plane along its ‘Z axis’. In the most common case, the animator selects the frequency channel. From left to right, the tape deck controls allow the user to:

- **rewind** to the start of the sequence (i.e., the first plane)
- **step backwards** by one plane
- **play backwards**, or repetitively step backwards
- **stop** any current play
- **play forward**, or repetitively step forward
- **step forward** by one plane
- **fast forward** to the end of the sequence

To the right of the tape deck is an editable text box indicating the current frame (channel) number and a label showing the total number of frames. Below that is a slider for controlling the (nominal) animation speed. To the right is a ‘Full/Compact’ toggle. In ‘Full’ mode (the default), a slider controlling frame number and a ‘Blink mode’ control are also available.

‘Blink’ mode is useful when more than one raster image is registered. In that mode, the tapedeck controls *which image* is displayed at the moment rather than the particular image plane (set that in ‘Normal’ mode first). The registered images must cover the same portion of the sky and use the same coordinate projection.

Note: *In ‘Normal’ mode, it is advisable to have only ONE raster image registered at a time, to avoid confusion. Unregister (or close) the others.*

At the bottom of the Display Panel is the **Position Tracking** panel. As the mouse moves over the main display, this panel shows information such as flux density, position (e.g. RA and Dec), Stokes,

and frequency (or velocity), for the point currently under the cursor. Each registered image/MS displays its own tracking information. Tracking can be 'frozen' (and unfrozen again) with the space bar. (Click on the main display area first, to be sure the keyboard is 'focused' there).

The Animator or Tracking panels can be hidden or detached (and later re-attached) by using the boxes at upper right of the panels; this is useful for increasing the size of the display area. (Use the 'View' menu to show a hidden panel again). The individual tracking areas (one for each registered image) can be hidden using the checkbox at upper left of each area.

7.2.2 Saving and Restoring Display Panel State

It is straightforward to save a display panel's current state (what data is on display along with data and panel settings). Select 'Save' (the unadorned floppy toolbutton) and confirm the filename. It is usually advisable (but not required) to retain the file's '.rstr' extension.

Press 'Restore' (the button to the right of Save) to choose a previously-created restore file. You can also select restore files from the Load Data window.

It is possible to restore MSs or images, multiple layers such as contour-over-raster, and LEL displays. You can also save the panel state with no data loaded, to restore preferred initial settings such as overall panel size. Animation and zoom state should likewise restore themselves.

Restore is fairly forgiving about data location, and will find files located:

- in the original location recorded in the restore file
- in the current working directory (where you started the viewer)
- in the restore file's directory
- in the original location relative to the restore file

This means that restore files will generally work if moved together with data files. The process is less forgiving if you save the display of an LEL (image) expression, however; the files must be in the locations specified in the original LEL expression. If a data file is not found, restore will attempt to proceed but results will vary.

Restore files are in ascii (xml) format, and some obvious manual edits are possible. However, these files are longer and more complex than you might imagine. Use caution, and back up restore files you want to preserve. If you make a mistake, the viewer may not recognize the file as a restore file; other unexpected results could also occur. It is usually easier and safer to make changes on the display panel and then save the restore file again.

7.2.3 Region Selection and Positioning

You can draw regions or select positions on the display with the mouse, once you have selected the appropriate tool(s) on the **Mouse Toolbar** (see above).

The **Rectangle Region** drawing tool currently works for the following:

- Region statistics reporting for images,
- Region spectral profiles for images, via the `Tools:Spectral Profile` menu,
- Flagging of Measurement Sets
- Creating and Saving an image region for various types of analysis (§ 7.3.5)
- Selecting Clean regions interactively (§ 5.3.5)

The `Polygon Region` drawing has the same uses, except that polygon region flagging of an MS is not supported.

The `Positioning` crosshair tool works for the last two of the above.

The `Spectral Profile` display (see § 7.3.4), when active, updates on *each change* of the rectangle, polygon, or crosshair. Flagging with the crosshair also responds to single click or drag.

Region statistics are printed in the terminal window (not the logger) by double-clicking the completed region. The `Rectangle Region` tool's mouse button must also be double-clicked to confirm an MS flagging edit.

Here is an example of region statistics from the viewer:

```
ngc5921.usecase.clean.image-contour      (Jy/beam)

n          Std Dev      RMS          Mean          Variance      Sum
52         0.01067      0.02412     0.02168      0.0001139    1.127

Flux      Med |Dev|     IntQtlRng   Median        Min           Max
0.09526   0.009185    0.01875    0.02076      0.003584     0.04181
```

7.2.4 The Load Data Panel

You can use the `Load Data - Viewer` GUI to interactively choose images or MS to load into the viewer. An example of this panel is shown in Figure 7.5. This panel is accessed through the `Data:Open` menu or `Open` icon of the `Viewer Display Panel`. It also appears if you open the viewer without any `infile` specified.

Selecting a file on disk in the `Load Data` panel will provide options for how to display the data. Images can be displayed as:

1. Raster Image,
2. Contour Map,
3. Vector map, or
4. Marker Map.

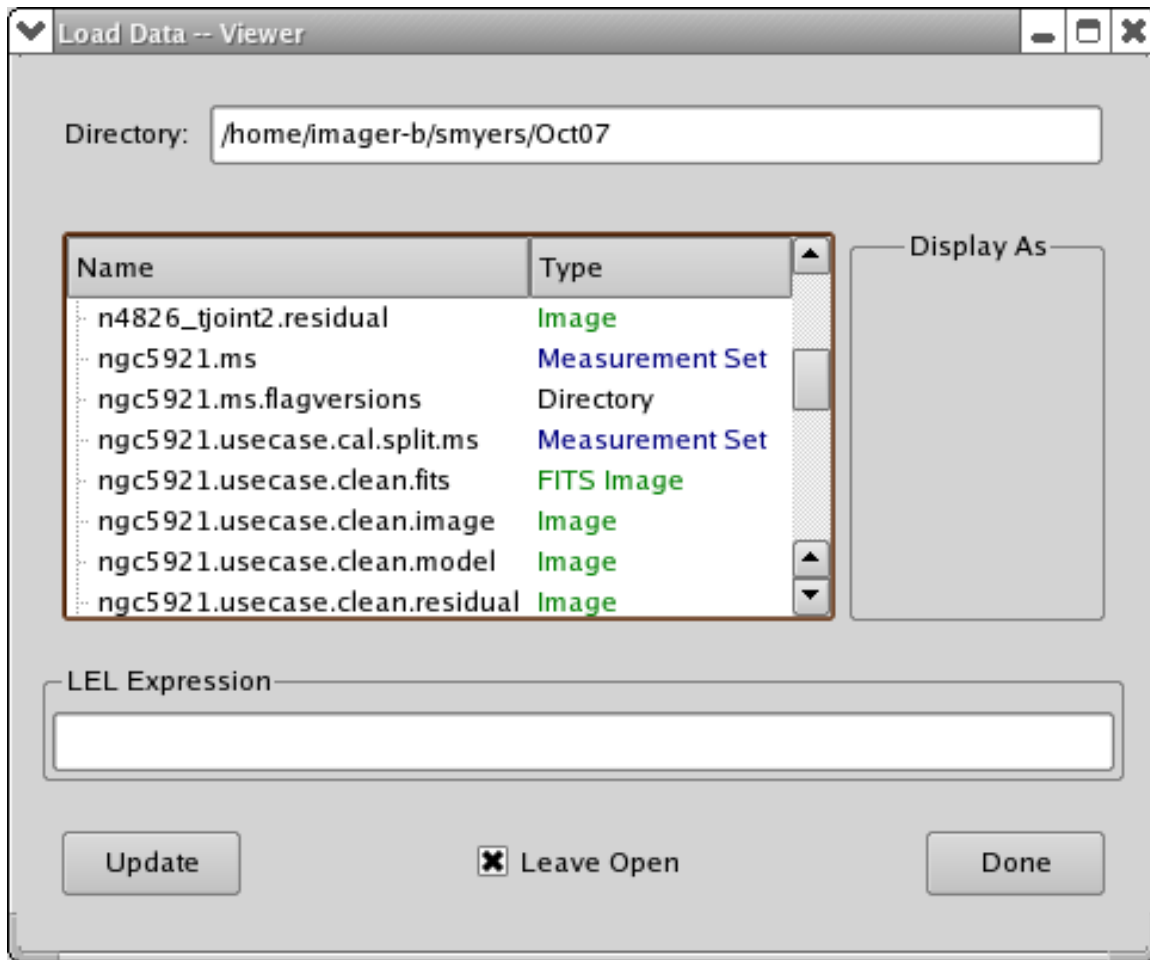


Figure 7.5: The **Load Data - Viewer** panel that appears if you open the viewer without any `infile` specified, or if you use the `Data:Open` menu or Open icon. You can see the images and MS available in your current directory, and the options for loading them.

You can also enter a 'Lattice Expression Language' (LEL) expression in the box provided (§ 6.1.3). For example, you might enter:

```
'my.clean.im' - 'my.dirty.im'
```

to display the difference between the two images. (The images should have the same coordinates and extents).

A MS can only be displayed as a raster.

7.2.4.1 Registered vs. Open Datasets

When you 'load' data as described above, it is first *opened*, and then *registered* on all existing **Display Panels**. The distinction is subtle. An 'open' dataset has been prepared in memory from disk; it may be registered (enabled for drawing) on one **Display Panel** and not on another. All open datasets will have a tab in the **Data Options** window, whether currently registered or not. On the other hand, only those datasets registered on a particular panel will show in its **Tracking** area.

At present, it is useful to have more than one image registered on a panel *only* if you are displaying a contour image over a raster image (§ 7.3.3) or 'blinking' between images (see **Animator** in § 7.2.1). (In future we also hope to provide transparent overlay of raster images).

It is the user's responsibility – and highly advisable – to unregister (or close) datasets that are no longer in use, using the **Register** or **Close** toolbutton or menu. In future the viewer will attempt to aid in unregistering datasets which are not 'compatible' with a newly-loaded one (different sky area, e.g., or MS vs. image).

If you close a dataset, you must reload it from disk as described above to see it again. That can take a little time for MSs, especially. If you unregister a dataset, it is set to draw immediately when you re-register it, with its options as you have previously set them. In general, close unneeded datasets but unregister those you'll be working with again.

7.3 Viewing Images

You have several options for viewing an image. These are seen at the right of the **Load Data - Viewer** panel described in § 7.2.4 and shown in Figure 7.6 when an image is selected. They are:

- **Raster Image** — a greyscale or color image,
- **Contour Map** — contours of intensity as a line plot,
- **Vector Map** — vectors (as in polarization) as a line plot,
- **Marker Map** — a line plot with symbols to mark positions.

The **Raster Image** is the default image display, and is what you get if you invoke the **viewer** from **casapy** with an image file name. In this case, you will need to use the **Open** menu to bring up the **Load Data** panel to choose a different display.

7.3.1 Viewing a raster map

A raster map of an image shows pixel intensities in a two-dimensional cross-section of gridded data with colors selected from a finite set of (normally) smooth and continuous colors, i.e., a colormap.

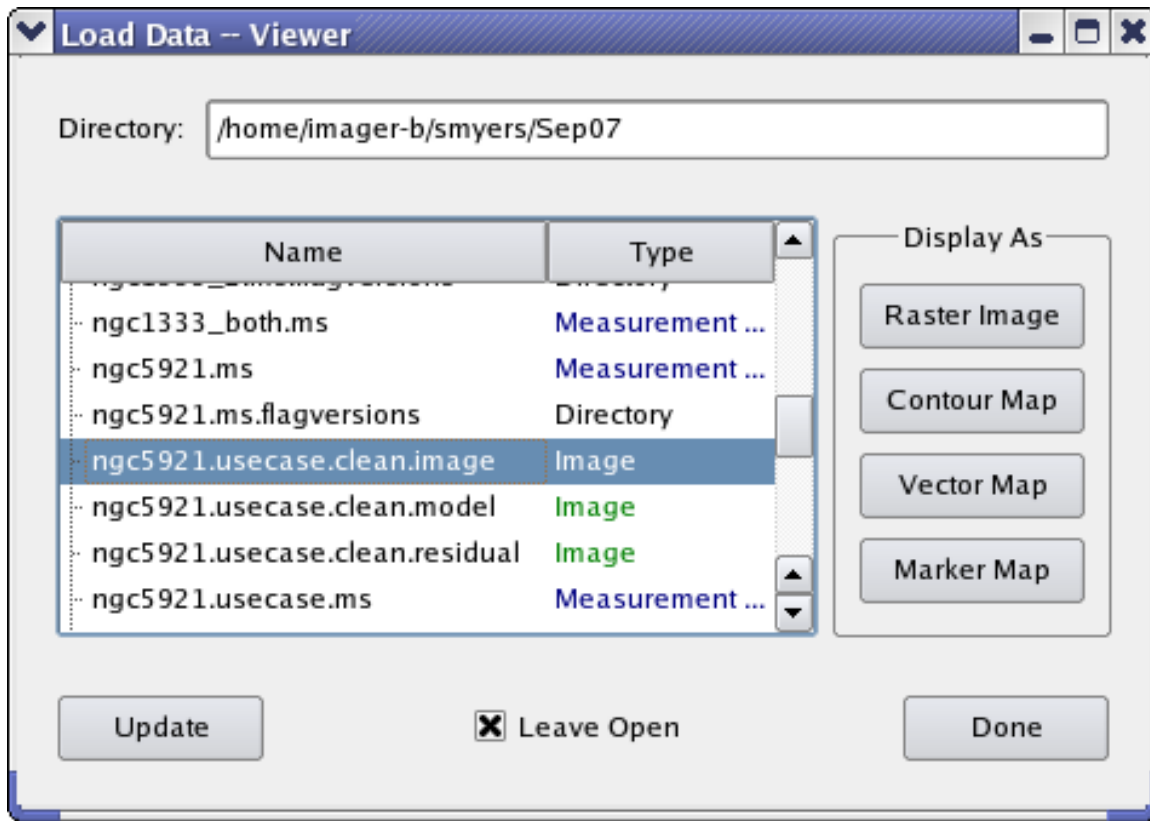


Figure 7.6: The **Load Data - Viewer** panel as it appears if you select an image. You can see all options are available to load the image as a **Raster Image**, **Contour Map**, **Vector Map**, or **Marker Map**. In this example, clicking on the **Raster Image** button would bring up the displays shown in Figure 7.1.

Starting the `casaviewer` with an image as a raster map will look something like the example in Figure 7.1.

You will see the GUI which consists of two main windows, entitled "Viewer Display Panel" and "Load Data". In the "Load Data" panel, you will see all of the viewable files in the current working directory along with their type (Image, Measurement Set, etc). After selecting a file, you are presented with the available display types (raster, contour, vector, marker) for these data. Clicking on the button **Raster Map** will create a display as above.

The data display can be adjusted by the user as needed. This is done through the **Data Display Options** panel. This window appears when you choose the `Data:Adjust` menu or use the wrench icon from the **Main Toolbar**. This also comes up by default along with the **Viewer Display Panel** when the data is loaded.

The **Data Display Options** window is shown in the right panel of Figure 7.1. It consists of a tab for each image or MS loaded, under which are a cascading series of expandable categories. For an

image, these are:

- **Display axes**
- **Hidden axes**
- **Basic Settings**
- **Position tracking**
- **Axis labels**
- **Axis label properties**
- **Beam Ellipse**
- **Color Wedge**

The **Basic Settings** category is expanded by default. To expand a category to show its options, click on it with the left mouse button.

7.3.1.1 Raster Image — Display Axes

In this category the physical axes (i.e. Right Ascension, Declination, Velocity, Stokes) to be displayed can be selected and assigned to the x, y, and z axes of the display. If your image has a fourth axis (typically Stokes), it can be controlled by a slider within the **Hidden axes** drop-down.

7.3.1.2 Raster Image — Basic Settings

This roll-up is open by default. It has some commonly-used parameters that alter the way the image is displayed; three of these affect the colors used. An example of this part of the panel is shown in Figure 7.7.

The options available are:

- **Basic Settings:Aspect ratio**

This option controls the horizontal-vertical size ratio of data pixels on screen. **Fixed world** (the default) means that the aspect ratio of the pixels is set according to the coordinate system of the image (i.e., true to the projected sky). **Fixed lattice** means that data pixels will always be square on the screen. Selecting **flexible** allows the map to stretch independently in each direction to fill as much of the display area as possible.

- **Basic Settings:Pixel treatment**

This option controls the precise alignment of the edge of the current 'zoom window' with the data lattice. **edge** (the default) means that whole data pixels are always drawn, even on the edges of the display. For most purposes, **edge** is recommended. **center** means that data

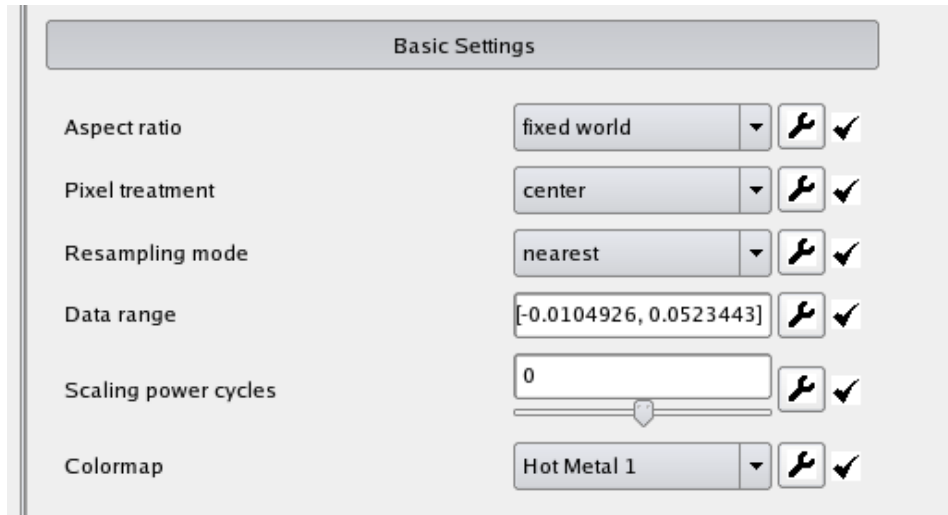


Figure 7.7: The **Basic Settings** category of the **Data Display Options** panel as it appears if you load the image as a **Raster Image**. This is a zoom-in for the data displayed in Figure 7.1.

pixels on the edge of the display are drawn only from their centers inwards. (Note that a data pixel's center is considered its 'definitive' position, and corresponds to a whole number in 'data pixel' or 'lattice' coordinates).

- **Basic Settings: Resampling mode**

This setting controls how the data are resampled to the resolution of the screen. **nearest** (the default) means that screen pixels are colored according to the intensity of the nearest data point, so that each data pixel is shown in a single color. **bilinear** applies a bilinear interpolation between data pixels to produce smoother looking images when data pixels are large on the screen. **bicubic** applies an even higher-order (and somewhat slower) interpolation.

- **Basic Settings: Data Range**

You can use the entry box provided to set the minimum and maximum data values mapped to the available range of colors as a list **[min, max]**. For very high dynamic range images, you will probably want to enter a **max** less than the data maximum in order to see detail in lower brightness-level pixels. The next setting also helps very much with high dynamic range data.

- **Basic settings: Scaling power cycles**

This option allows logarithmic scaling of data values to colormap cells.

The color for a data value is determined as follows: first, the value is clipped to lie within the data range specified above, then mapped to an index into the available colors, as described in the next paragraph. The color corresponding to this index is determined finally by the current colormap and its 'fiddling' (shift/slope) and brightness/contrast settings (see **Mouse**

Toolbar, above). Adding a **Color Wedge** to your image can help clarify the effect of the various color controls.

The `Scaling power cycles` option controls the mapping of clipped data values to colormap indices. Set to zero (the default), a straight linear relation is used. For negative scaling values, a logarithmic mapping assigns an larger fraction of the available colors to lower data values (this is usually what you want). Setting `dataMin` to something around the noise level is often useful/appropriate in conjunction with a negative 'Power cycles' setting.

For positive values, an larger fraction of the colormap is used for the high data values.¹

See Figure 7.8 for sample curves.

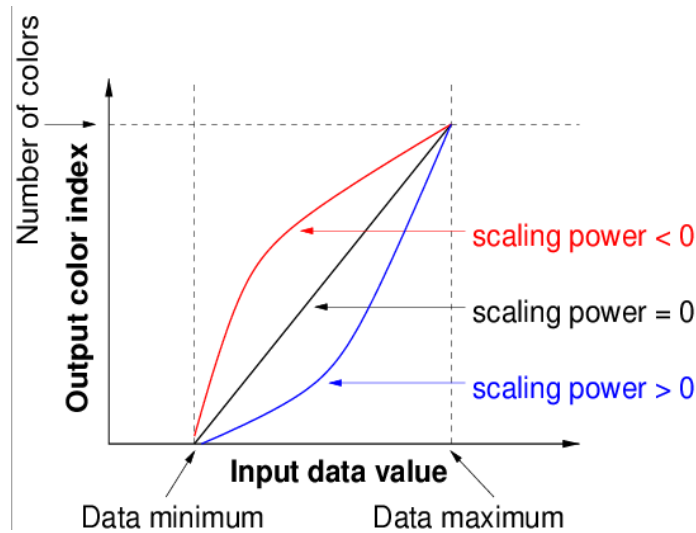


Figure 7.8: Example curves for scaling power cycles.

- **Basic settings:** Colormap

You can select from a variety of colormaps here. `Hot Metal`, `Rainbow` and `Greyscale` colormaps are the ones most commonly used.

¹The actual functions are computed as follows:

For negative scaling values (say $-p$), the data is scaled linearly from the range (`dataMin` – `dataMax`) to the range $(1 - 10^p)$. Then the program takes the log (base 10) of that value (arriving at a number from 0 to p) and scales that linearly to the number of available colors. Thus the data is treated as if it had p decades of range, with an equal number of colors assigned to each decade.

For positive scaling values, the inverse (exponential) functions are used. If p is the (positive) value chosen, The data value is scaled linearly to lie between 0 and p , and 10 is raised to this power, yielding a value in the range $(1 - 10^p)$. Finally, that value is scaled linearly to the number of available colors.

7.3.1.3 Raster Image — Other Settings

Many of the other settings on the `Data Options` panel for raster images are self-explanatory, such as those which affect `Beam ellipse` drawing (only available if your image provides beam data), or the form of the `Axis labeling` and `Position tracking` information. You can also give your image a `Color wedge`, a key to the current mapping from data values to colors.

You can control which of your image’s axes are on the vertical and horizontal display axes and which on the animation or ‘movie’ axis, within the `Display axes` drop-down. You must set the X, Y and Z (animation) axes so that each shows a *different* image axis, in order for your choice to take effect.

7.3.2 Viewing a contour map

Viewing a contour image is similar to the process above. A contour map shows lines of equal data value (e.g., flux density) for the selected plane of gridded data (Figure 7.9). Contour maps are particularly useful for overlaying on raster images so that two different measurements of the same part of the sky can be shown simultaneously (§ 7.3.3).

Several `Basic Settings` options control the contour levels used. The contours themselves are specified by a list in the `RelativeContourLevels` box. These are defined relative to the two other parameters, the `BaseContourLevel` (which sets what 0 in the relative contour list corresponds to in the image), and the `UnitContourLevel` (which sets what 1 in the relative contour list corresponds to in the image). Note that negative contours are usually dashed. **ALERT:** This scheme was adopted in 2.4.0 and is slightly different to that used in previous versions.

For example, it is relatively straightforward to set fractional contours (e.g. “percent levels”), e.g.:

```
RelativeContourLevels = [0.2, 0.4, 0.6, 0.8]
BaseContourLevel = 0.0
UnitContourLevel = <image max>
```

This maps the maximum to 1 and thus our contours are fractions of the peak.

Another example shows how to set absolute values so that the contours are given in flux density units (Jy):

```
RelativeContourLevels = [0.010, 0.020, 0.040, 0.080, 0.160, 0.320]
BaseContourLevel = 0.0
UnitContourLevel = 1.0
```

Here we have contours starting at 10mJy and doubling every contour.

We can also set contours in multiples of the image rms (“sigma”):

```
RelativeContourLevels = [-3,3,5,10,15,20]
BaseContourLevel = 0.0
UnitContourLevel = <image rms>
```

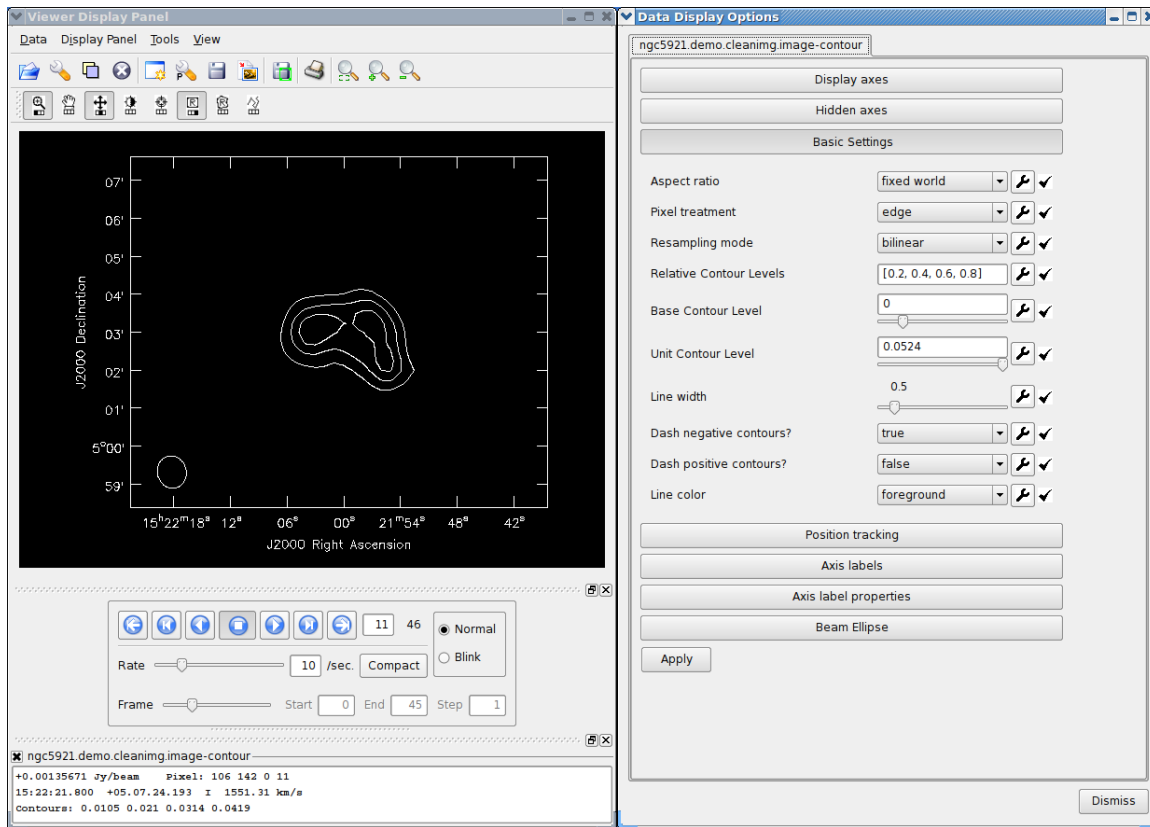



Figure 7.9: The **Viewer Display Panel** (left) and **Data Display Options** panel (right) after choosing **Contour Map** from the **Load Data** panel. The image shown is for channel 11 of the NGC5921 cube, selected using the **Animator** tape deck, and zoomed in using the tool bar icon. Note the different options in the open **Basic Settings** category of the **Data Display Options** panel.

Here we have first contours at negative and positive 3-sigma. You can get the image rms using the `imstat` task (§ 6.7) or using the Viewer statistics tool on a region of the image (§ 7.2.3).

As a final example, not all images are of intensity, for example a moment-1 image (§ 6.6) has units of velocity. In this case, absolute contours will work fine, but by default the viewer will set fractional contours but referred to the min and max velocity:

```
RelativeContourLevels = [0.2, 0.4, 0.6, 0.8]
BaseContourLevel = <image min>
UnitContourLevel = <image max>
```

Here we have contours spaced evenly from min to max, and this is what you get by default if you load a non-intensity image (like the moment-1 image). See Figure 7.10 for an example of this.

7.3.3 Overlay contours on a raster map

Contours of either a second data set or the same data set can be used for comparison or to enhance visualization of the data. The Data Options Panel will have multiple tabs which allow adjusting each overlay individually (Note tabs along the top). **Beware:** it's easy to forget which tab is active! Also note that **axis labeling** is controlled by the *first-registered* image overlay that has labeling turned on (whether raster or contour), so make label adjustments within that tab.

To add a Contour overlay, open the **Load Data** panel (Use the **Data** menu or click on the Folder icon), select the data set and select **Contour**. See Figure 7.10 for an example using NGC5921.

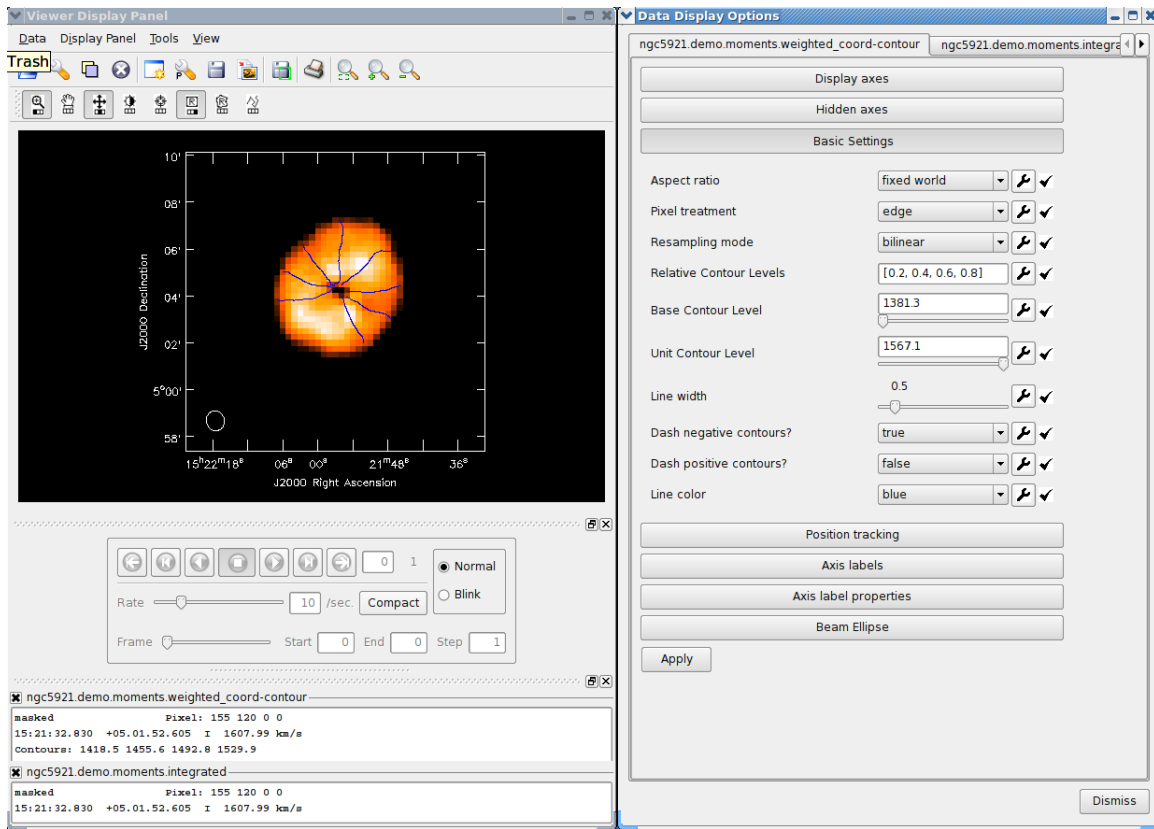


Figure 7.10: The **Viewer Display Panel** (left) and **Data Display Options** panel (right) after overlaying a **Contour Map** of velocity on a **Raster Image** of intensity. The image shown is for the moments of the NGC5921 cube, zoomed in using the tool bar icon. The tab for the contour plot is open in the **Data Display Options** panel.

7.3.4 Spectral Profile Plotting

From the **Tools** menu, the **Spectral Profile** plotting tool can be selected. This will pop up a new **Image Profile** window containing an x-y plot of the intensity versus spectral axis (usually

velocity). You can then select a region with the **Rectangle** or **Polygon Region** drawing tools, or pinpoint a position using the **Crosshair** tool. The profile for the region or position selected will then appear in the **Image Profile** window. This profile will update in real time to track changes to the region or crosshair, which can be moved by click-dragging the mouse. See Figure 7.11.

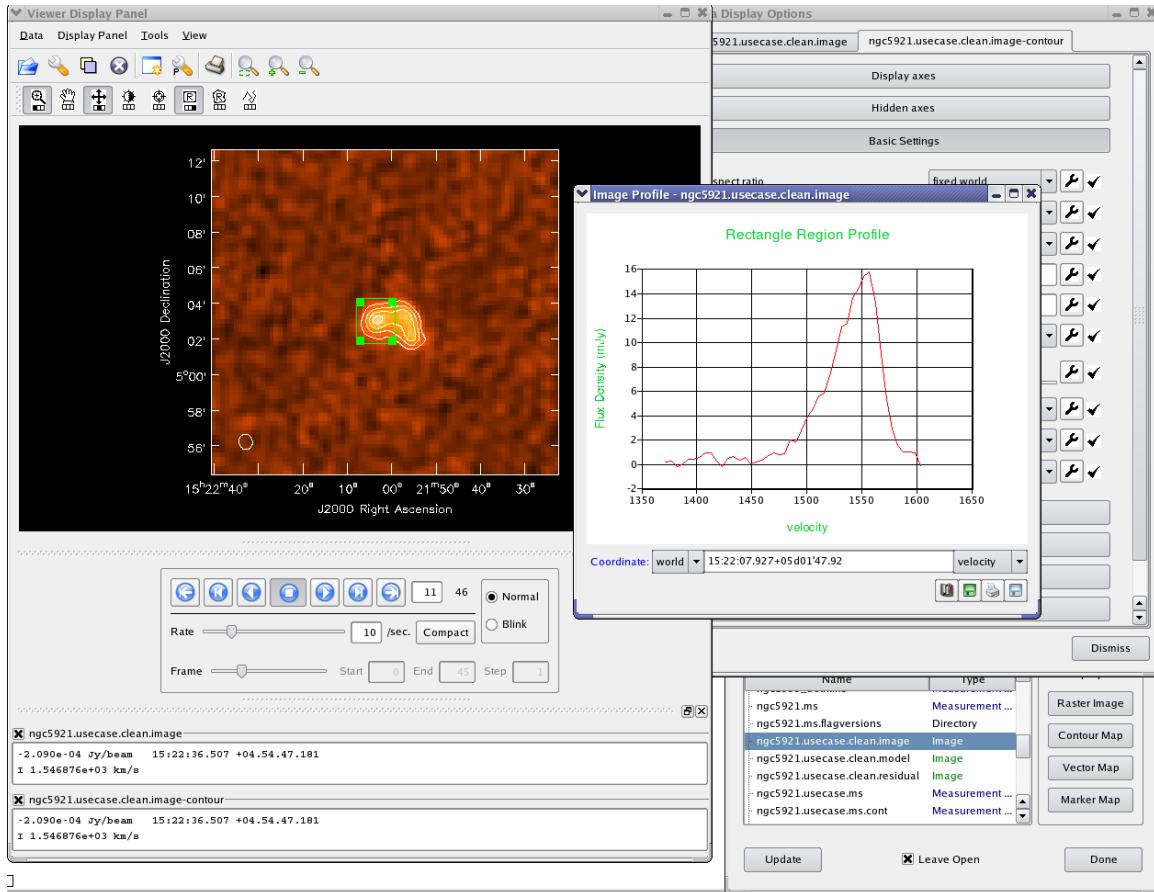


Figure 7.11: The **Image Profile** panel that appears if you use the **Tools:Spectral Profile** menu, and then use the rectangle or polygon tool to select a region in the image. You can also use the crosshair to get the profile at a single position in the image. The profile will change to track movements of the region or crosshair if moved by dragging with the mouse.

7.3.5 Managing and Saving Regions

Regions can be saved in different ways in CASA **Alert: the variety of regions/boxes/masks may be reduced in future releases**. Apart from the mask images (i.e. images that contain 0 and 1 values for False and True respectively), there are ascii box (*.box) files and binary region (*.rgn) files.

First, trace out your region on the display panel using the rectangle or polygon region mouse tools (§ 7.2.1, § 7.2.3), and confirm by double-clicking inside the region. Figure 7.12 shows an image region selected with the polygon tool.

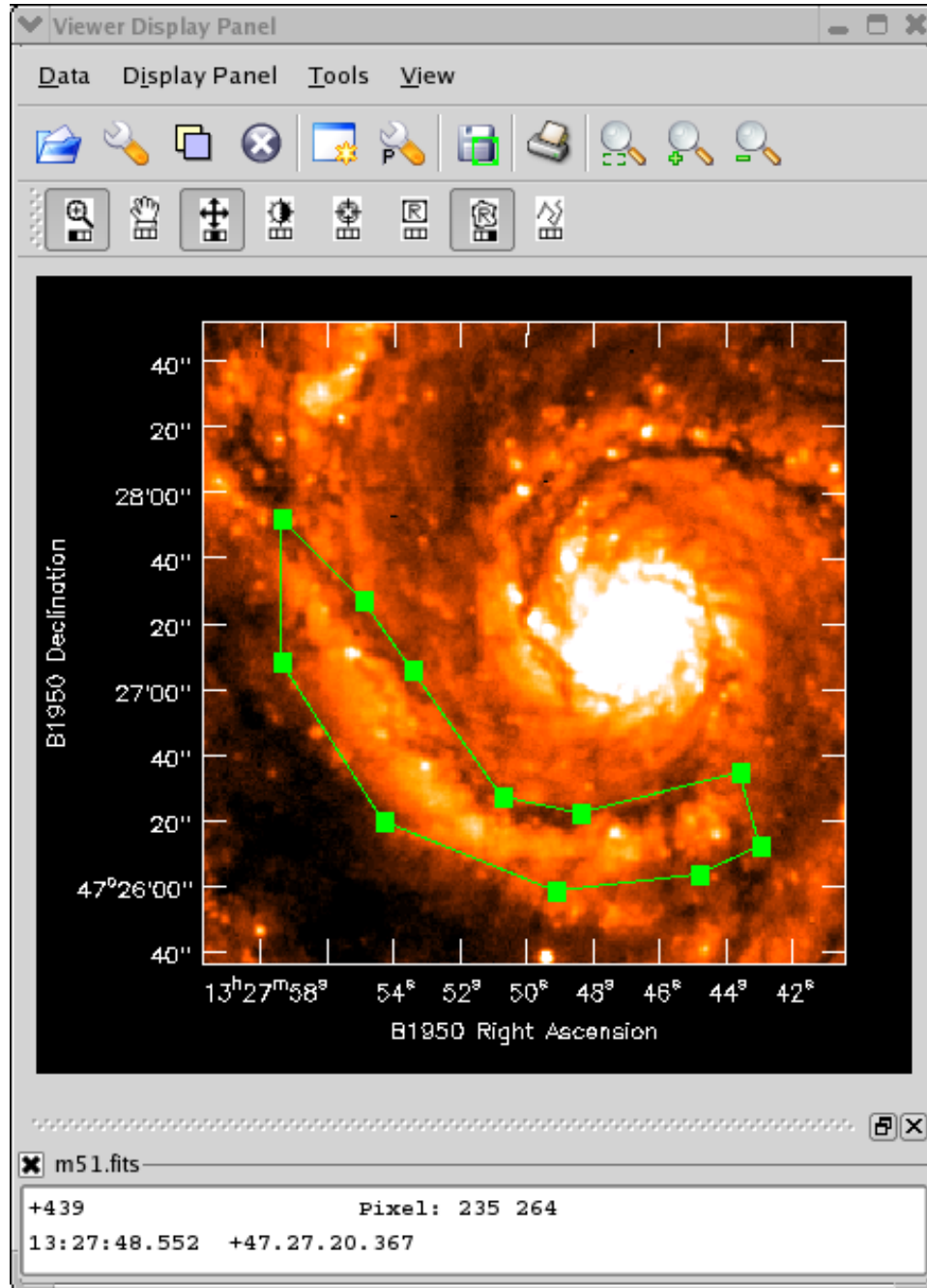


Figure 7.12: Selecting an image region with the polygon tool.

Then open one of the following pop-up windows via the **Tools** menu:

- **Box in File** This option will produce an ascii file that can be stored in any place on your hard drive.
- **Region in File** This option will produce a binary file that can be stored in any place on your hard drive. The binary file can also be used in other CASA tasks.
- **Region in Image** As the region above, but the region file itself will be stored within the image. No separate file is produced.

All three options will be controlled by a pop-up window similar to the one shown in Fig.7.13. **Channels** and **Stokes** lets you expand the region onto those two axes, effectively making the region file three or four-dimensional. They use the usual selection syntax. **Load** and **Save** buttons will read and write a file in the requested format (for the **Region in Image** an internal name can be selected instead). **CleanUp** will remove all regions, **Hide/Show** are toggles to display the regions. Finally a color can be selected.

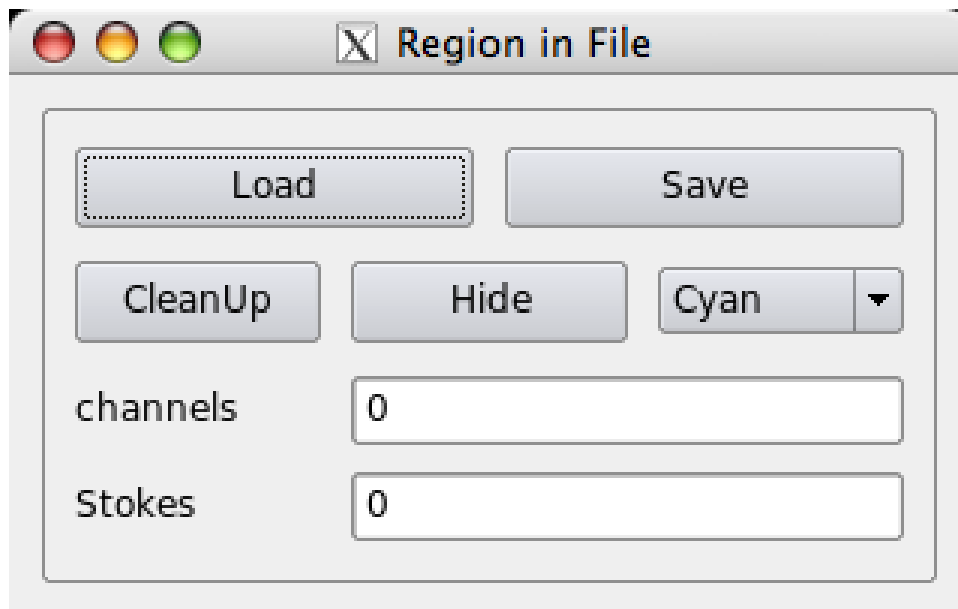


Figure 7.13: A typical pop-up that appears for the selection of regions and boxes

Alert: The **Shape Manager** in the **Tools** menu, serves to place annotations on the images. Its use is experimental at this stage.

7.3.6 Adjusting Canvas Parameters/Multi-panel displays

The display area can also be manipulated with the following controls in the **Panel Options** (or 'Viewer Canvas Manager') window. Use the wrench icon with a 'P' (or the 'Display Panel' menu)

to show this window.

- Margins - specify the spacing for the left, right, top, and bottom margins
- Number of panels - specify the number of panels in x and y and the spacing between those panels.
- Background Color - white or black (more choices to come)

7.3.6.1 Setting up multi-panel displays

Figure 7.14 illustrates a multi-panel display along with the Viewer Canvas Manager settings which created it.

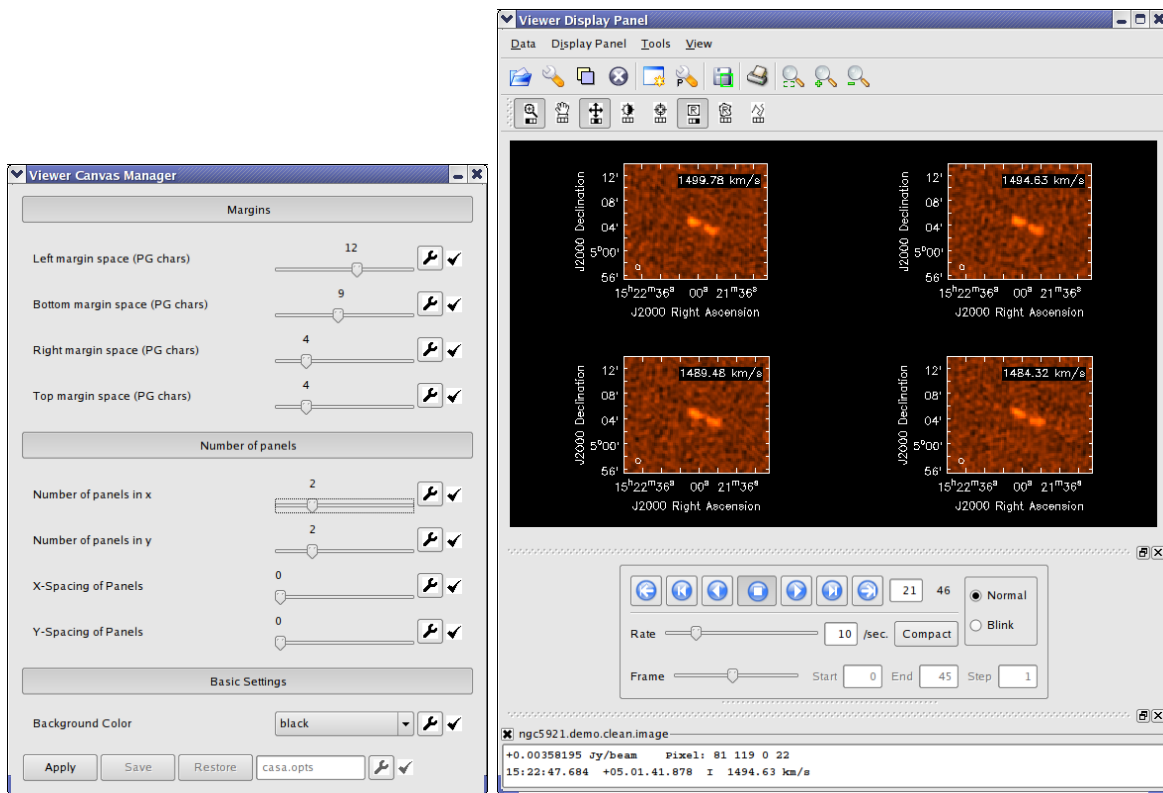


Figure 7.14: A multi-panel display set up through the **Viewer Canvas Manager**.

7.3.6.2 Background Color

The **Background Color** selection can be used to change the background color from its default of **black**. Currently, the only other choice is **white**, which is more appropriate for printing or inclusion in documents.

7.4 Viewing Measurement Sets

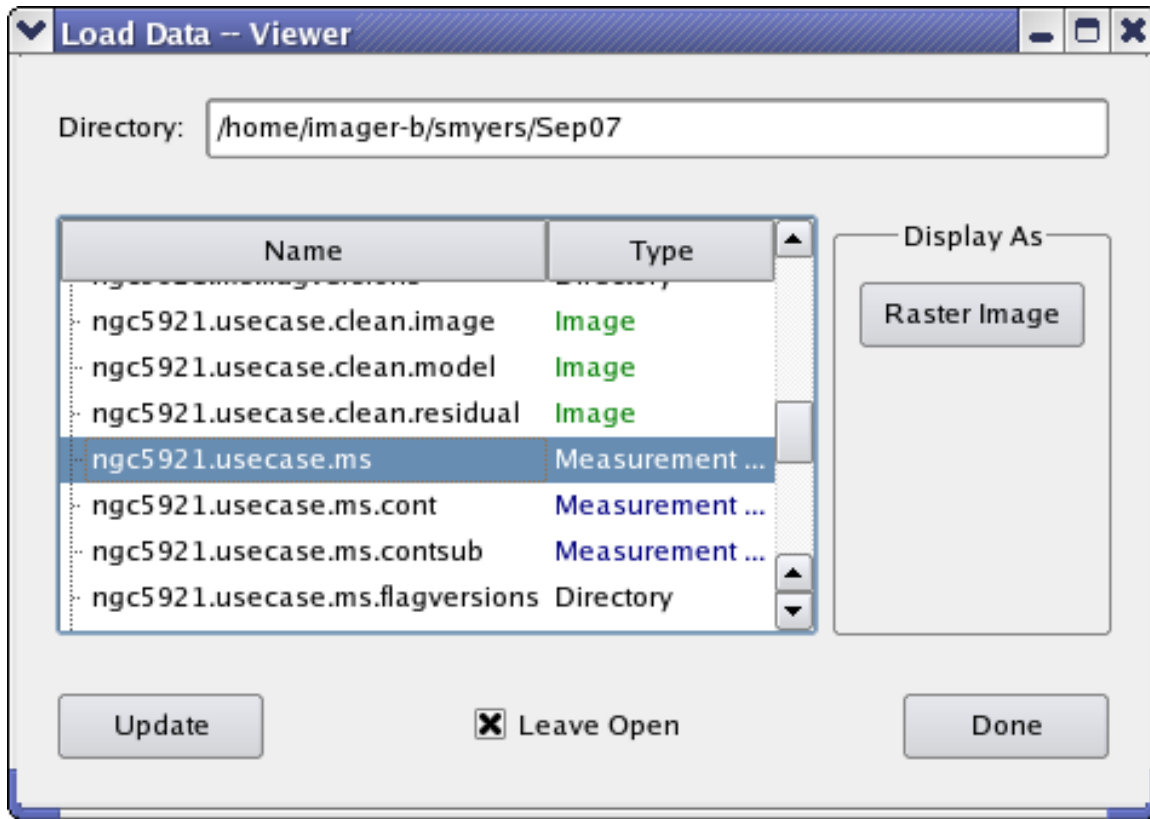


Figure 7.15: The **Load Data - Viewer** panel as it appears if you select an MS. The only option available is to load this as a **Raster Image**. In this example, clicking on the **Raster Image** button would bring up the displays shown in Figure 7.2.

Visibility data can also be displayed and flagged directly from the viewer. For Measurement Set files the only option for display is 'Raster' (similar to AIPS task TVFLG). An example of MS display is shown in Figure 7.2; loading of an MS is shown in Figure 7.15.

Warning: *Only one MS should be registered at a time on a Display Panel.* Only one MS can be shown in any case. You do not have to close other images/MSs, but you should at least 'unregister' them from the Display Panel used for viewing the MS. If you wish to see other images or MSs at the same time, create multiple Display Panel windows.

7.4.1 Data Display Options Panel for Measurement Sets

The **Data Display Options** panel provides adjustments for MSs similar to those for images, and also includes flagging options. As with images, this window appears when you choose the

Data:Adjust menu or use the wrench icon from the **Main Toolbar**. It is also shown by default when an MS is loaded. The right panel of Figure 7.2 shows a **Data Options** window. It has a tab for each open MS, containing a set of categories. The options within each category can be either 'rolled up' or expanded by clicking the category label.

For a Measurement Set, the categories are:

- **Advanced**
- **MS and Visibility Selection**
- **Display Axes**
- **Flagging Options**
- **Basic Settings**
- **Axis Drawing and Labels**
- **Color Wedge**

7.4.1.1 MS Options — Basic Settings

The **Basic Settings** roll-up is expanded by default. It contains entries similar to those for a raster image (§ 7.3.1.2). Together with the brightness/contrast and colormap adjustment icons on the **Mouse Toolbar** of the Display Panel, they are especially important for adjusting the color display of your MS.

The available Basic options are:

- **Data minimum/maximum**
This has the same usage as for raster images. Lowering the data maximum will help brighten weaker data values.
- **Scaling power cycles**
This has exactly the same usage as for raster images (see § 7.3.1.2). Again, lowering this value often helps make weaker data visible. If you want to view several fields with very different amplitudes simultaneously, this is typically one of the best adjustments to make early, together with the **Colormap fiddling** mouse tool, which is on the middle mouse button by default.
- **Colormap**
Greyscale or **Hot Metal** colormaps are generally good choices for MS data.

7.4.1.2 MS Options— MS and Visibility Selections

- Visibility Type
- Visibility Component
- Moving Average Size

This roll-up provides choice boxes for Visibility Type (Observed, Corrected, Model, Residual) and Component (Amplitude, Phase, Real, or Imaginary).

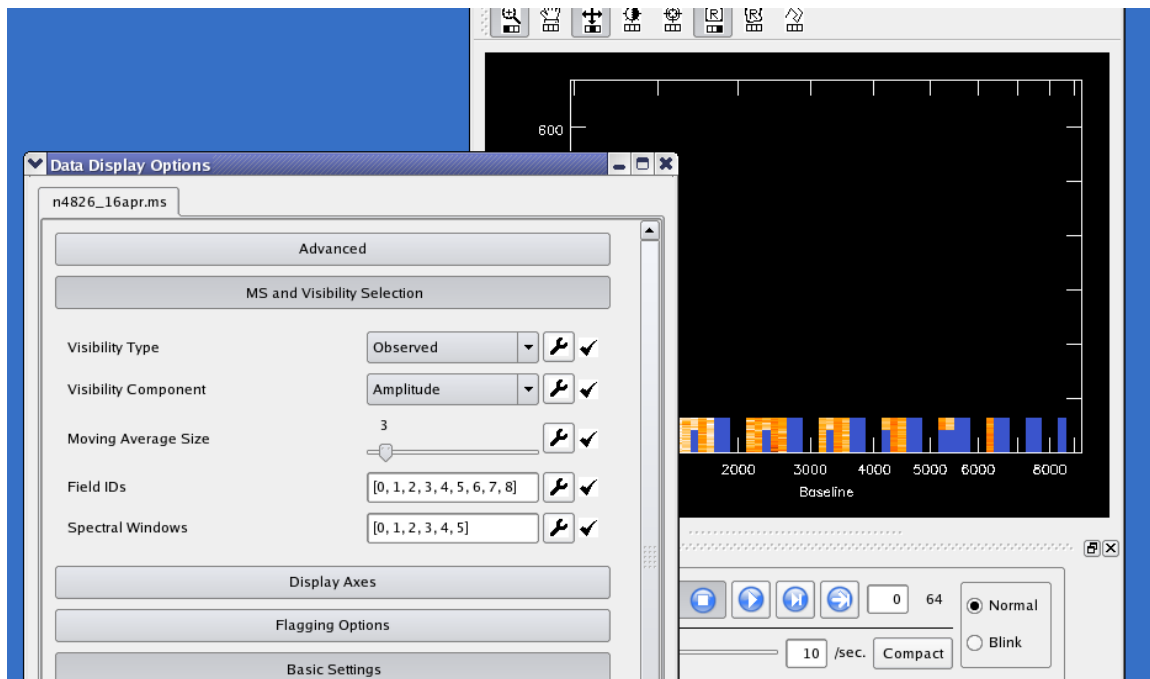


Figure 7.16: The MS for NGC4826 BIMA observations has been loaded into the viewer. We see the first of the `spw` in the Display Panel, and have opened up `MS and Visibility Selections` in the **Data Display Options** panel. The display panel raster is not full of visibilities because `spw 0` is continuum and was only observed for the first few scans. This is a case where the different spectral windows have different numbers of channels also.

Changes to Visibility Type or Component (changing from Phase to Amplitude, for example) require the data to be retrieved again from the disk into memory, which can be a lengthy process. When a large MS is first selected for viewing, the user must trigger this retrieval manually by pressing the **Apply** button (located below all the options), after selecting the data to be viewed (see `Field IDs` and `Spectral Windows`, below).

Tip: Changing visibility type between 'Observed' and 'Corrected' can also be used to assure that data and flags are reloaded from disk. You should do this if you're using another flagging tool

such as autoflag simultaneously, so that the viewer sees the other tool's new edits and doesn't overwrite them with obsolete flags. The **Apply** button alone won't reload unless something within the viewer itself requires it; in the future, a button will be provided to reload flags from the disk unconditionally.

You can also choose to view the difference from a running mean or the local RMS deviation of either Phase or Amplitude. There is a slider for choosing the nominal number of time slots in the 'local neighborhood' for these displays.

(Note: **Insufficient Data** is shown in the tracking area during these displays when there is no other unflagged data in the local neighborhood to compare to the point in question. The moving time windows will not extend across changes in either field ID or scan number boundaries, so you may see this message if your scan numbers change with every time stamp. An option will be added later to ignore scan boundaries).

- Field IDs
- Spectral Windows

You can retrieve and edit a selected portion of the MS data by entering the desired Spectral Window and Field ID numbers into these boxes. **Important:** Especially with large MSs, often the first thing you'll want to do is to select **spectral windows** which all have the **same number of channels** and the **same polarization setup**. It also makes sense to edit only a few fields at a time. Doing this will also greatly reduce data retrieval times and memory requirements.

You can separate the ID numbers with spaces or commas; you do not need to enter enclosing brackets. Changes to either entry box will cause the selected MS data to be reloaded from disk.

If you select, say, spectral windows 7, 8, 23, and 24, the animator, slice position sliders, and axis labeling will show these as 0, 1, 2, and 3 (the 'slice positions' or 'pixel coordinates' of the chosen spectral windows). Looking at the position tracking display is the best way to avoid confusion in such cases. It will show something like: **Sp Win 23 (s 2)** when you are viewing spectral window 23 (plane 2 of the selected spectral windows).

Changes to MS selections will not be allowed until you have saved (or discarded) any previous edits you have made (see **Flagging Options -- Save Edits**, below). A warning is printed on the console (not the logger).

Initially, all fields and spectral windows are selected. To revert to this 'unselected' state, choose 'Original' under the wrench icons next to the entry boxes.

See Figure 7.16 for an example showing the use of the **MS and Visibility Selections** controls when viewing an MS.

7.4.1.3 MS Options — Display Axes

This roll-up is very similar to that for images: it allows the user to choose which axes (from Time, Baseline, Polarization, Channel, and Spectral Window) are on the display and the animator.

There are also sliders here for choosing positions on the remaining axes. (It's useful to note that the data *is* actually stored internally in memory as an array with these five axes).

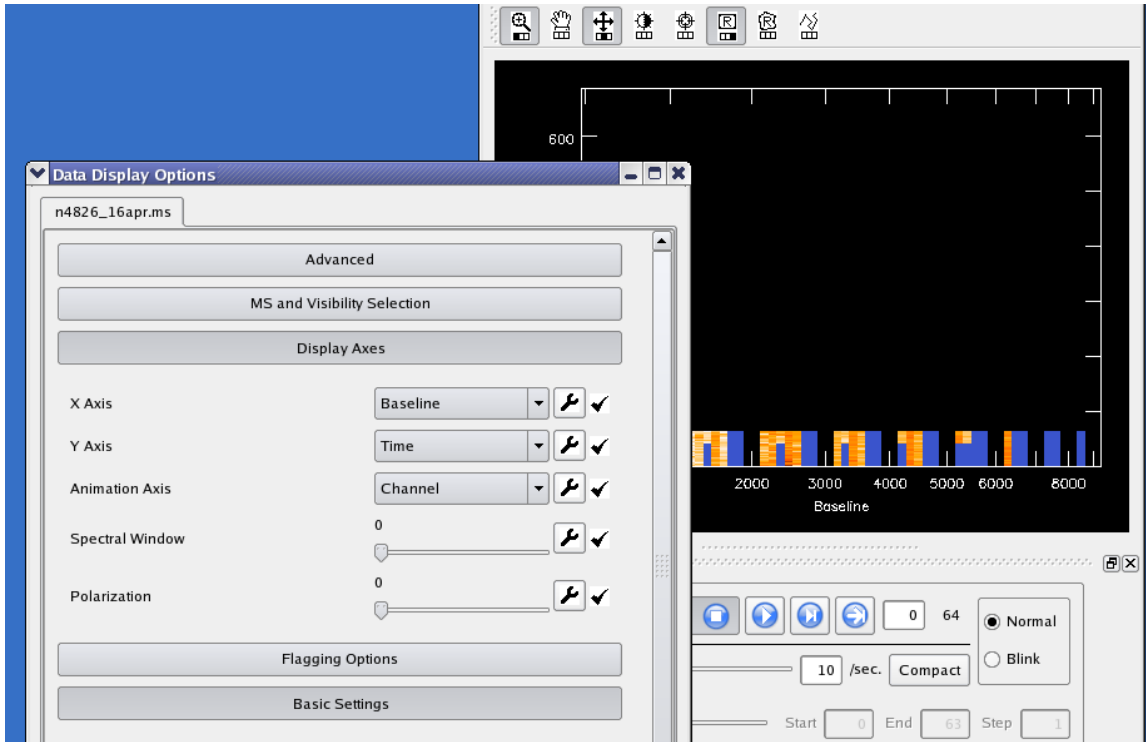


Figure 7.17: The MS for NGC4826 from Figure 7.16, now with the **Display Axes** open in the **Data Display Options** panel. By default, channels are on the **Animation Axis** and thus in the tapedeck, while spectral window and polarization are on the **Display Axes** sliders.

For MSs, changing the choice of axis on one control will automatically swap axes, maintaining different axes on each control. Changing axes or slider/animator positions does not normally require pressing **Apply** — the new slice is shown immediately. However, the display may be partially or completely grey in areas if the required data is not currently in memory, either because no data has been loaded yet, or because not all the selected data will fit into the allowed memory. Press the **Apply** button in this case to load the data (see § 7.4.1.6 and **Max. Visibility Memory** at the end of § 7.4.1.5).

Within the **Display Axes** rollup you may also select whether to order the baseline axis by antenna1-antenna2 (the default) or by (unprojected) baseline length.

See Figures 7.17–7.18 showing the use of the **Display Axes** controls to change the axes on the animation and sliders.

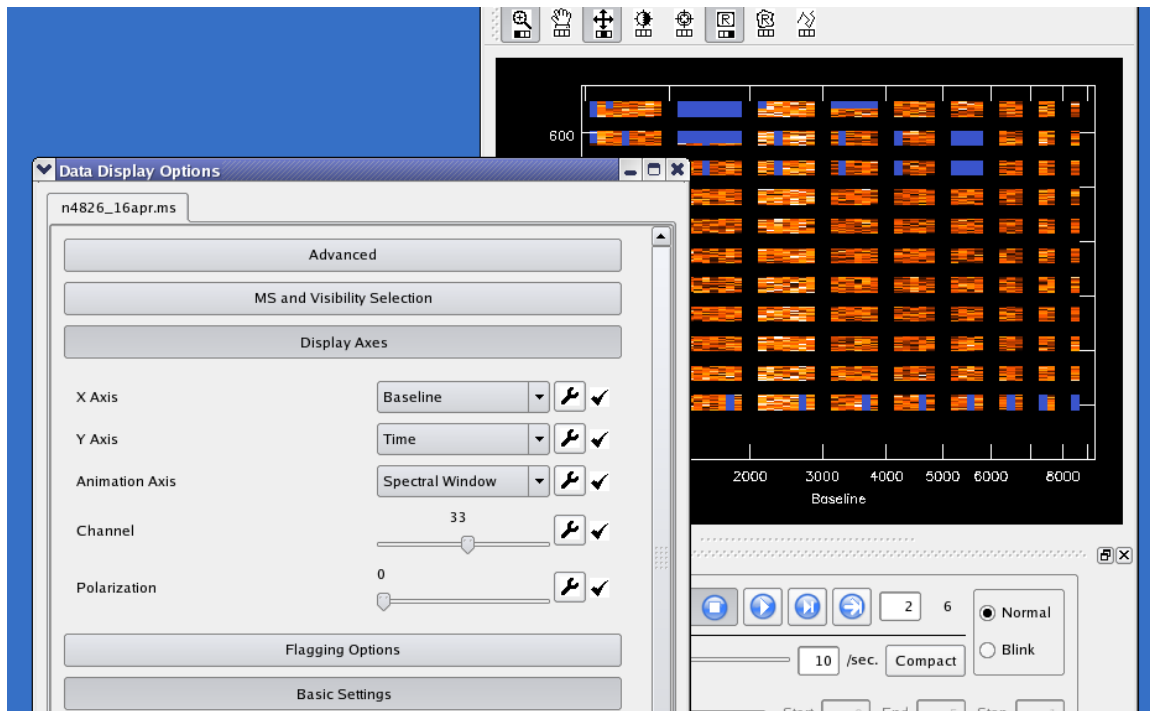


Figure 7.18: The MS for NGC4826, continuing from Figure 7.17. We have now put `spectral window` on the **Animation Axis** and used the tape deck to step to `spw 2`, where we see the data from the rest of the scans. Now `channels` is on a **Display Axes** slider, which has been dragged to show `Channel 33`.

7.4.1.4 MS Options — Flagging Options

These options allow you to edit (flag or unflag) MS data. The **Crosshair and Rectangle Region Mouse Tools** (§ 7.2.3) are used on the display to select the area to edit. When using the **Rectangle Region** tool, double-click inside the selected rectangle to confirm the edit.

The options below determine how edits will be applied.

- **Show Flagged Regions...**

You have the option to display flagged regions in the background color (as in TVFLG) or to highlight them with color. In the former case, flagged regions look just like regions of no data. With the (default) color option, flags are shown in shades of blue: darker blue for flags already saved to disk, lighter blue for new flags not yet saved; regions with no data will be shown in black.

- **Flag or Unflag**

This setting determines whether selected regions will be flagged or unflagged. This does *not*

affect previous edits; it only determines the effect which later edits will have. Both flagging and unflagging edits can be accumulated and then saved in one pass through the MS.

- **Flag/Unflag All...**

These flagging extent checkboxes allow you to extend your edit over any of the five data axes. For example, to flag *all* the data in a given time range, you would check all the axes *except* Time, and then select the desired time range with the **Rectangle Region** mouse tool. Such edits will extend along the corresponding axes over the entire selected MS (whether loaded into memory or not) and optionally over unselected portions of the MS as well (**Use Entire MS**, below). Use care in selecting edit extents to assure that you're editing all the data you wish to edit.

- **Flag/Unflag Entire Antenna?**

This control can be used to extend subsequent edits to all baselines which include the desired antenna[s]. For example, if you set this item to 'Yes' and then click the crosshair on a visibility point with baseline 3-19, the edit would extend over baselines 0-3, 1-3, 2-3, 3-3, 3-4, ... **3-nAntennas-1**. Note that the second antenna of the selection (19) is irrelevant here – you can click anywhere within the 'Antenna 3 block', i.e., where the *first* antenna number is 3, to select all baselines which include antenna 3.

This item controls the edit extent only along the baseline axis. If you wish to flag *all* the data for a given antenna, you must still check the boxes to flag all Times, Channels, Polarizations and Spectral Windows. There would be no point, however, in activating *both* this item and the 'Flag All Baselines' checkbox. You can flag an antenna in a limited range of times, etc., by using the appropriate checkboxes and selecting a rectangular region of visibilities with the mouse.

Note: You do not need to include the entire 'antenna block' in your rectangle (and you may stray into the next antenna if you try). Anywhere within the block will work. To flag higher-numbered antennas, it often helps to zoom in.

- **Undo Last Edit**

- **Undo All Edits**

The 'Undo' buttons do the expected thing: completely undo the effect of the last edit (or all unsaved edits). Please note, however, that only unsaved edits can be undone here; there is no ability to revert to the flagging state at the start of the session once flags have been saved to disk (unless you have previously saved a 'flag version'. The flag version tool is not available through the viewer directly).

- **Use Entire MS When Saving Edits?**

"Yes" means that saving the edits will flag/unflag over the entire MS, *including* fields (and possibly spectral windows) which are not currently selected for viewing. Specifically, data within time range(s) you swept out with the mouse (even for unselected fields) will be edited.

In addition, if "Flag/Unflag All..." boxes were checked, such edits will extend throughout the MS. Note that only unselected *times* (fields) can be edited *without* checking extent boxes for

the edits as well. Unselected spectral windows, e.g., will *not* be edited unless the edit also has "Flag/Unflag All Spectral Windows" checked.

Warning: Beware of checking "All Spectral Windows" unless you have also checked "All Channels" or turned "Entire MS" off; channel edits appropriate to the selected spectral windows may not be appropriate to unselected ones. Set "Use Entire MS" to "No" if your edits need to apply only to the portion of the MS you have selected for viewing. *Edits can often be saved significantly faster this way as well.*

Also note that checkboxes apply to individual edits, and must be checked before making the edit with the mouse. "Use Entire MS", on the other hand, applies to all the edits saved at one time, and must be set as desired before pressing "Save Edits".

- **Save Edits**

MS editing works like a text editor in that you see all of your edits immediately, but nothing is committed to disk until you press 'Save Edits'. Feel free to experiment with all the other controls; nothing but 'Save Edits' will alter your MS on disk. As mentioned previously, however, there is no way to undo your edits once they are saved, except by manually entering the reverse edits (or restoring a previously-saved 'flag version').

Also, *you must save* (or discard) *your edits before changing the MS selections*. If edits are pending, the selection change will not be allowed, and a warning will appear on the console.

If you close the MS in the viewer, *unsaved edits are simply discarded*, without prior warning. It's important, therefore, to remember to save them yourself. You can distinguish unsaved flags (when using the 'Flags In Color' option), because they are in a lighter shade of blue.

The program must make a pass through the MS on disk to save the edits. This can take a little time; progress is shown in the console window.

7.4.1.5 MS Options— Advanced

These settings can help optimize your memory usage, especially for large MSs. A rule of thumb is that they can be increased until response becomes sluggish, when they should be backed down again.

You can run the unix 'top' program and hit 'M' in it (to sort by memory usage) in order to examine the effects of these settings. Look at the amount of RSS (main memory) and SWAP used by the X server and 'casaviewer' processes. If that sounds familiar and easy, then fiddling with these settings is for you. Otherwise, the default settings should provide reasonable performance in most cases.

- **Cache size**

The value of this option specifies the maximum number of different views of the data to save so that they can be redrawn quickly. If you run an animation or scroll around zoomed data, you will notice that the data displays noticeably faster the second time through because of this feature. Often, setting this value to the number of animation frames is ideal. Note, however, that on multi-panel displays, each panel counts as one cached image.

Large images naturally take more room than small ones. The memory used for these images will show up in the X server process. If you need more Visibility Memory (below) for a really large ms, it is usually better to forgo caching a large number of views.

- **Max. Visibility Memory**

This option specifies how many megabytes of memory may be used to store visibility data from the measurement set internally. *Even if you do not adjust this entry, it is useful to look at it to see how many megabytes are required to store your entire (selected) MS in memory.* If the slider setting is above this, the whole selected MS will fit into the memory buffer. Otherwise, some data planes will be 'greyed out' (see **Apply Button**, § 7.4.1.6 below), and the selected data will have to be viewed one buffer at a time, which is somewhat less convenient. In most cases, this means you should **select fewer fields or spectral windows** – see § 7.4.1.2. The 'casaviewer' process contains this buffer memory (it contains the entire viewer, but the memory buffer can take most of the space).

7.4.1.6 MS Options — Apply Button

When viewing large MSs the display may be partially or completely grey in areas where the required data is not currently in memory, either because no data has been loaded yet, or because not all the selected data will fit into the allowed memory (see **Max. Visibility Memory** above). When the cursor is over such an area, the following message shows in the position tracking area:

```
press 'Apply' on Adjust panel to load data
```

Pressing the **Apply** button (which lies below all the options) will reload the memory buffer so that it includes the slice you are trying to view.

The message **No Data** has a different meaning; in that case, there simply *is* no data in the selected MS at the indicated position.

For large measurement sets, loading visibility data into memory is the most time-consuming step. Progress feedback is provided in the console window. Again, careful selection of the data to be viewed can greatly speed up retrieval.

7.5 Printing from the Viewer

You can use the **Data:Print** menu or the **Print** button to bring up the **Viewer Print Manager**. From this panel, you can print a hardcopy of what is in the Display Panel, or save it in a variety of formats.

Figure 7.19 shows an example of printing to a file. The key to making acceptable hardcopies (particularly for printing or inclusion in documents) is to set the background color and line widths to appropriate values so the plot and labels show up in the limited resolution of the hardcopy.

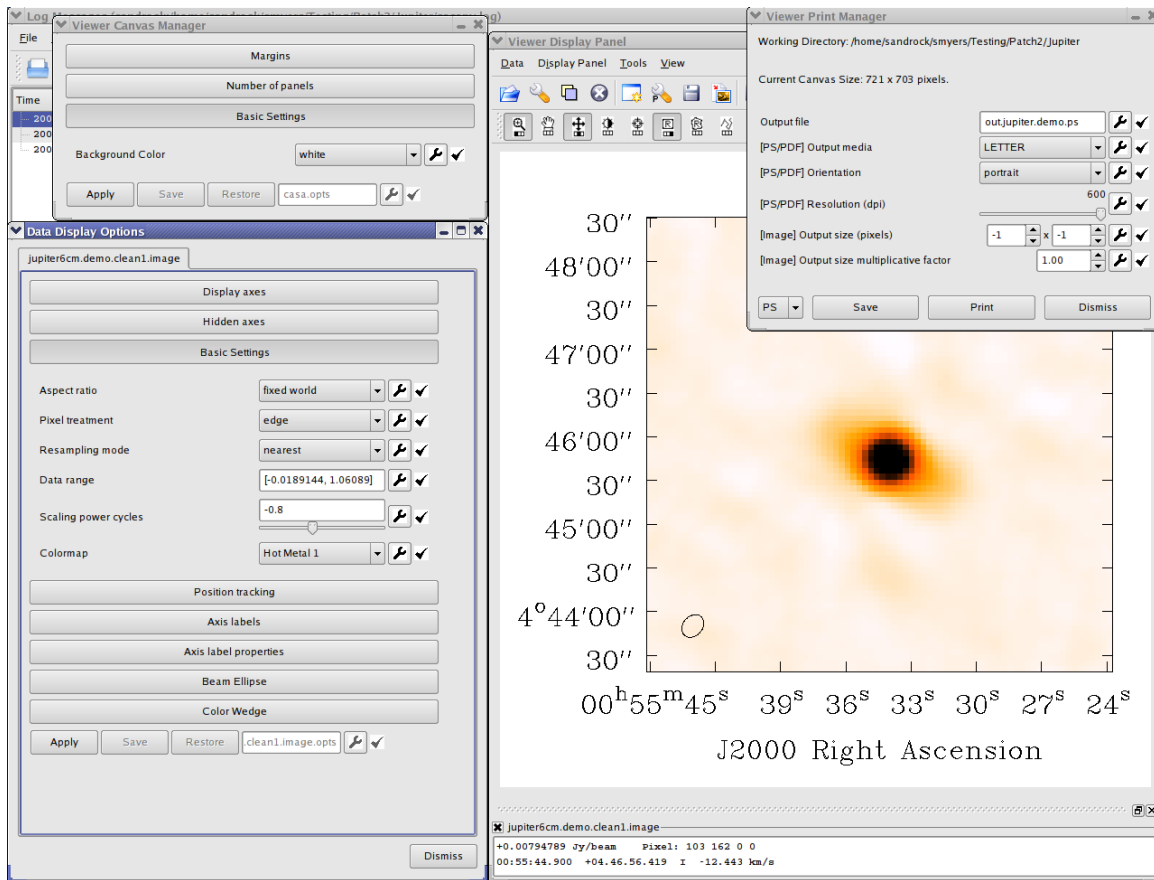


Figure 7.19: Setting up to print to a file. The background color has been set to **white**, the line width to 2, and the print resolution to 600 dpi (for an postscript plot). To make the plot, use the **Save** button on the **Viewer Print Manager** panel (positioned in the figure in the upper right) and select a format with the drop-down, or use the **Print** button to send directly to a printer.

Use the **Viewer Canvas Manager** (§ 7.3.6) to change the **Background Color** from its default of **black** to **white** if you are making plots for printing or inclusion in a document. You might also want to change the **colormap** accordingly.

Adjust the **Line Width** of the **Axis Label Properties** options in the **Data Display Options** panel so that the labels will be visible when printed. Increasing from the default of 1.4 to a value around 2 seems to work well.

You can choose an output file name in the panel. Be sure to make it a new name, otherwise it will not overwrite a previous file (and will not say anything about it).

If you will be printing to a postscript printer or to a PS or EPS file, dial up the **[PS] Resolution (dpi)** to its maximum of 600. This will increase the size of the PS file somewhat, but will make a much better plot. Use **gzip** to compress the PS file if necessary. Be sure to choose the desired

Output Media and Orientation for PS also.

BETA ALERT: The postscript printing capabilities of the `casaviewer` are currently limited due to some issues with the way we use Qt and do axis labels. Significant improvements have been made in Patch 3. This will be further upgraded in the future, but for now you will need to follow the suggestions above to get a useable plot. Note that `ghostview` may show a poorer version of the PS than you will get when you print.

Appendix A

Single Dish Data Processing

ALERT: The single-dish analysis package within CASA is still experimental. It is included in the release for the use of the ALMA computing and commissioning groups. It is not intended for general users. Therefore, this description is included in this Cookbook as an appendix.

For single-dish spectral calibration and analysis, CASA uses the ATNF Spectral Analysis Package (ASAP). ASAP is imported as the `sd` tool via

```
asap_init()
```

It forms the basis for a series of tasks (the “SDtasks”) that encapsulates the functionality within the standard CASA task framework. ASAP was developed to support the Australian telescopes such as Mopra, Parkes, and Tidbinbilla, and we have adapted it for use within CASA for GBT (see the note below for limitation of GBT SDFITS handling) and eventually ALMA [Note: Some support for the ALMA is now available]. **For R3.0.1, the ASAP version included in CASA was updated to Version 2.3.1 (which is the latest official release of ASAP as of Mar. 2010).** For details on ASAP – including the User Guide, Reference Manual, and tutorial – see the ASAP home page at ATNF:

- <http://www.atnf.csiro.au/computing/software/asap/> .

The ASAP tools are prefaced with `sd.` within CASA, e.g., the ASAP tool `scantable` becomes `sd.scantable`. See § A.3 for more information on the tools.

All of the ASAP functionality is available within the CASA installation. Since we extended ASAP, there are certain functionalities that are available only in the CASA version of ASAP. In the following sections of this appendix, we outline how to access ASAP from within CASA and the data flow for standard use cases.

If you run into trouble, be sure to check the list of known issues and features of ASAP and the SDtasks presented in § A.5 first.

An important note for GBT raw SDFITS data: The GBT raw SDFITS data format (in which the data are stored in multiple data tables within SDFITS file) are not supported in the current SDtasks or sd toolkit. Thus the data reduction for the GBT data starting from the raw SDFITS cannot be done. It is generally possible to process the GBT data once it is converted with tools outside CASA to a single data table SDFITS or other data format supported by ASAP.

A.1 Guidelines for Use of ASAP and SDtasks in CASA

A.1.1 Environment Variables

There are a number of environment variables that the ASAP tools (and thus the SDtasks) use to control their operation. They are located in `.asaprc` and are described in the ASAP User Guide. Within CASA, they are contained in the Python dictionary `sd.rcParams` and are accessible through its keys and values. For SDtask users, the most important parameter is `verbose`, which controls the display of detailed messages from the tools. By default,

```
sd.rcParams['verbose'] = True
```

produced by lots of messages. Also, the `scantable.storage` parameter controls whether scantable operations are done in memory or on disk. The default is

```
sd.rcParams['scantable.storage'] = 'memory'
```

which is the best choice if there is enough memory does it in memory. On the other hand,

```
sd.rcParams['scantable.storage'] = 'disk'
```

forces the task to store datasets on disk, which might be necessary when they are large. See § A.3.1 for more details on the ASAP environment variables.

A.1.2 Assignment

Some ASAP methods and functions require assigning a method to a variable which can then be manipulated. These methods and functions include `sd.scantable` and `sd.selector`, both of which make objects. For example,

```
s = sd.scantable('OrionS_rawACSmod', average=False)
```

A.1.3 Lists

For lists of scans or IFs, such as in `scanlist` and `iflist` in the SDtasks, the tasks and functions require a comma-delimited Python list, e.g.,

```
scanlist = [241, 242, 243, 244, 245, 246]
```

The python `range` function can be used to generate a list of consecutive numbers, e.g.,

```
scanlist = range(241,247)
```

giving the same list as above,

```
CASA <3>: scanlist=range(241,247)
CASA <4>: print scanlist
[241, 242, 243, 244, 245, 246]
```

Multiple ranges can be created by summing lists,

```
CASA <5>: scanlist=range(241,247) + range(251,255)
CASA <6>: print scanlist
[241, 242, 243, 244, 245, 246, 251, 252, 253, 254]
```

Note that in the future, the `sd` tools and `SDtasks` will use the same selection language as in the interferometric synthesis part of the `CASA`.

Spectral regions, such as those for setting masks, are pairs of min and max values for whatever spectral axis unit is currently chosen. These are fed into the tasks and tools as a list of lists, where each list element is a list with the `[min,max]` for that sub-region, e.g.,

```
masklist=[[1000,3000], [5000,7000]]
```

A.1.4 Dictionaries

Currently, the `SDtasks` return the Python dictionary for the results of line fitting (in `sdfit`) and region statistics (in `sdstat`). If you invoke these tasks by assigning a variable for the return, you can then access the elements through the keywords, e.g.,

```
CASA <10>: line_stat=sdstat()
Current fluxunit = K
No need to convert fluxunits
Using current frequency frame
Using current doppler convention
```

```
CASA <11>: line_stat
Out[11]:
{'eqw': 70.861755476162784,
 'max': 1.2750182151794434,
 'mean': 0.35996028780937195,
 'median': 0.23074722290039062,
 'min': -0.20840644836425781,
 'rms': 0.53090775012969971,
 'stddev': 0.39102539420127869,
 'sum': 90.350028991699219}
```

One can then use these values in scripts by accessing this dictionary, e.g.,

```
CASA <12>: print "Line max = %5.3f K" % (line_stat['max'])
Line max = 1.275 K
```

A.1.5 Line Formatting

The SDtasks trap leading and trailing whitespace on string parameters (such as `infile` and `sdfilename`) but ASAP does not, so be careful with setting string parameters. ASAP is case-sensitive, with most parameters being upper-case, such as `ASAP` for the `sd.scantable.save` file format. The SDtasks are generally more forgiving. Also, beware Python's sensitivity to indentation.

A.1.6 Logging

Before R3.0, all messages from ASAP were written to the standard output (`sys.stdout`) and they disappeared after exiting CASA. For R3.0, the logging system of ASAP is integrated into CASA logging system. Therefore, all outputs from ASAP commands, except for GUI related notifications, are sent to the file `casapy.log` and they are displayed to the CASA Logger (see § 1.4.2).

A.2 Single Dish Analysis Tasks

A set of single dish tasks is available for simplifying basic reduction activities. The list currently includes:

- **sdaverage** — select, calibrate, and average SD data
- **sdsMOOTH** — smooth SD spectra
- **sdbaseline** — fit/remove spectral baselines from SD data
- **sdcal** — `sdaverage`, `sdsMOOTH`, and `sdbaseline` combined to perform standard single dish processing all at once
- **sdcoadd** — merge/co-add multiple SD data
- **sdflag** — channel/row flagging of SD spectra
- **sdFIT** — line fitting to SD spectra
- **sdIMAGING** — create an image from the total power or spectral data
- **sdIMPROCESS** — remove the 'scanning noise' from raster scanned data
- **sdLIST** — print a summary of a SD dataset
- **sdMATH** — do simple arithmetic for SD spectra
- **sdPLOT** — plotting of SD spectra, including overlay of line catalog data
- **sdsAVE** — save SD data to different format
- **sdSCALE** — scale SD data

- **sdstat** — compute statistics of regions of SD spectra
- **sdtipimaging** — do a simple calibration and create an image from the total power raster scans
- **sdsim** — simulate single dish data (experimental)

All of the SDtasks, except those related to imaging (**sdtipimaging**, **sdimaging**, and **sdimprocess**), work from a file on disk rather than from a scantable in memory as the ASAP toolkit does (see § A.3. Inside the tasks we invoke a call to **sd.scantable** to read the data. The scantable objects do not persist within CASA after completion of the tasks and are destroyed to free up memory.

Three tasks **sdaverage**, **sdsmooth**, and **sdbaseline** are the workhorses for the calibration, selection, averaging, baseline fitting, and smoothing. The output datasets for each task are written to a file on disk. Alternatively, one can use the task **sdcal** to perform all of the steps in the three tasks described. Its operation is controlled by three main "mode" parameters: **calmode** (which selects the type of calibration, if any, to be applied), **kernel** (which selects the smoothing), and **blmode** (which selects baseline fitting). There are also parameters controlling the selection such as **scanlist**, **iflist**, **field**, **scanaverage**, **timeaverage**, and **polaverage**. Note that **sdcal** can be run with **calmode='none'** to allow re-selection or writing out of data that is already calibrated. There is a "wiring diagram" of the dataflow and control inputs for **sdcal** shown in Figure A.1.

The SDtasks support the import and export file formats supported by ASAP itself. For import, this includes: ASAP (scantables), MS (CASA measurement set), RPFITS and SDFITS. For export, this includes: ASAP (scantables), MS (CASA measurement set), ASCII (text file), SDFITS (a flavor of SD FITS). The **sdsave** task is available exclusively for exporting with these data selection options. The **sdcoadd** task is available to merge data in separate data files into one. A brief summary of the data in a file is found in the **sdlist** task help.

Plotting of spectra is handled in the **sdplot** task. It also offers some selection, averaging, and smoothing options in case you are working from a dataset that has not been split or averaged. Note that there is some rudimentary plotting capability in many of SD tasks, controlled through the **plotlevel** parameter, to aid in the assessment of the performance of these tasks.

Scaling of the spectra and Tsys is available in the **sdscale**. For arithmetic operations of spectra in separate scantables, a new task, **sdmath** has been added.

Calculation of statistics on spectral regions is available in the **sdstat** task. Results are passed by a Python dictionary return variable.

Basic Gaussian line-fitting is handled by the **sdfit** task. It can deal with the simpler cases, and offers some automation as well as interactive selection of fitting region, but more complicated fitting is best accomplished through the toolkit (**sd.fitter**).

Basic non-interactive channel and row flagging is available in the **sdflag** task. By default or by specifying outfile parameter, a new file is created containing dataset with the flag information. To update flags in the input data, **outfile='none'** must be set.

Limited total power data analysis functionality is available through the task **sdtipimaging**. A single dish image data cube can be created using **sdimaging**, which also handles total power imaging.

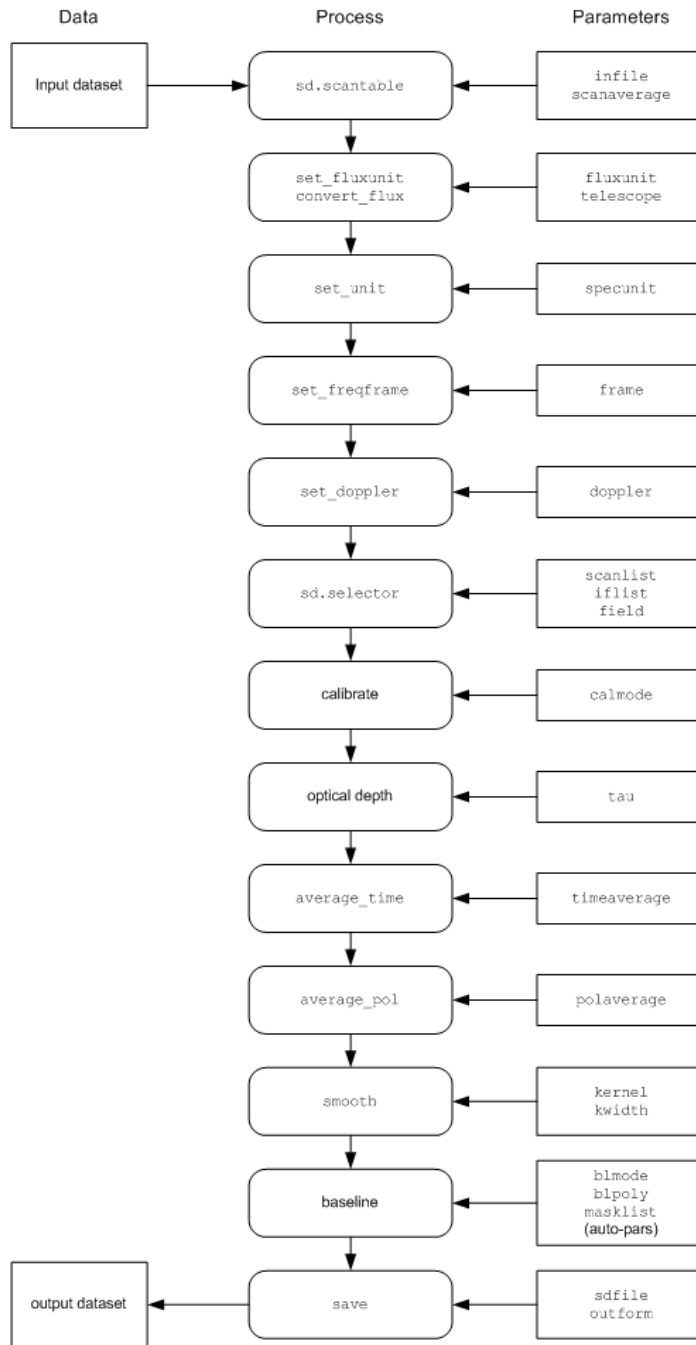


Figure A.1: Wiring diagram for the SDtask `sdcal`. The stages of processing within the task are shown, along with the parameters that control them.

These tasks directly access the Measurement Set without converting it to scantable format. The `sdimprocess` is intended to remove the 'scanning noise' from single dish images by either the

'Basket-Weaving' or 'Pressed-out' methods.

Although the Measurement Set can store data from multiple antennas even if it consists of only single-dish spectra (auto-correlation data), the scantable cannot distinguish data from multiple antennas. It causes a problem when the user processes the Measurement Set using SDtasks. Therefore, id or name of the antenna that the user want to process must be explicitly specified if the input dataset for SDtasks is Measurement Set. This can be done by `antenna` parameter. By default (`antenna=0`), data associate with antenna id 0 is imported. The `antenna` parameter takes no effect for other input data formats.

A.2.1 SDtask Summaries

The following are the list of parameters and brief descriptions of each of the SDtasks. These descriptions are also contained in `help <taskname>` once `asap_init` has been invoked.

A.2.1.1 sdaverage

Keyword arguments:

`sdfile` -- name of input SD dataset

`antenna` -- antenna name or id (only effective for MS input)

`fluxunit` -- units for line flux

options: 'K','Jy',''

default: '' (keep current fluxunit)

WARNING: For GBT data, see description below.

>>> `fluxunit` expandable parameter

`telescopeparm` -- the telescope characteristics

options: (str) name or (list) list of gain info

default: '' (none set)

example: if `telescopeparm=''`, it tries to get the telescope name from the data.

Full antenna parameters (diameter,ap.eff.) known to ASAP are

'ATPKSMB', 'ATPKSHOH', 'ATMOPRA', 'DSS-43',

'CEDUNA', 'HOBART'. For GBT, it fixes default fluxunit to 'K' first then converts to a new fluxunit.

`telescopeparm=[104.9,0.43]` diameter(m), ap.eff.

`telescopeparm=[0.743]` gain in Jy/K

`telescopeparm='FIX'` to change default fluxunit

see description below

`specunit` -- units for spectral axis

options: (str) 'channel','km/s','GHz','MHz','kHz','Hz'

default: '' (=current)


```

        example: these will be the units for masklist
frame -- frequency frame for spectral axis
  options: (str) 'LSRK','REST','TOPO','LSRD','BARY',
            'GEO','GALACTO','LGROUP','CMB'
  default: currently set to frame in scantable
  WARNING: frame='REST' not yet implemented
doppler -- doppler mode
  options: (str) 'RADIO','OPTICAL','Z','BETA','GAMMA'
  default: currently set to doppler in scantable
calmode -- calibration mode
  options: 'ps','nod','fs','fsotf','quotient','none'
  default: 'none'
  example: choose mode 'none' if
           already calibrated and want to
           try averaging
scanlist -- list of scan numbers to process
  default: [] (use all scans)
  example: [21,22,23,24]
           this selection is in addition to field,
           iflist, and pollist
field -- selection string for selecting scans by name
  default: '' (no name selection)
  example: 'FLS3a*'
           this selection is in addition to scanlist,
           iflist, and pollist
iflist -- list of IF id numbers to select
  default: [] (use all IFs)
  example: [15]
           this selection is in addition to scanlist,
           field, and pollist
pollist -- list of polarization id numbers to select
  default: [] (use all polarizations)
  example: [1]
           this selection is in addition to scanlist,
           field, and iflist
channelrange -- channel range selection
  default: [] (use all channels)
  example: [0,5000]
           Note that specified values are recognized as
           'channel' regardless of the value of specunit
scanaverage -- average integrations within scans
  options: (bool) True,False
  default: False
  example: if True, this happens in read-in
           For GBT, set False!

```

```

timeaverage -- average times for multiple scan cycles
  options: (bool) True,False
  default: False
  example: if True, this happens after calibration

>>>timeaverage expandable parameter
  tweight -- weighting for time average
    options: 'none'
            'var' (1/var(spec) weighted)
            'tsys' (1/Tsys**2 weighted)
            'tint' (integration time weighted)
            'tintsys' (Tint/Tsys**2)
            'median' ( median averaging)
    default: 'none'
  averageall -- average multi-resolution spectra
    spectra are averaged by referring
    their frequency coverage
    default: False

polaverage -- average polarizations
  options: (bool) True,False
  default: False

>>>polaverage expandable parameter
  pweight -- weighting for polarization average
    options: 'none'
            'var' (1/var(spec) weighted)
            'tsys' (1/Tsys**2 weighted)
    default: 'none'

tau -- atmospheric optical depth
  default: 0.0 (no correction)

verify -- verify the results of calibration. Only effective if
  calmode is not 'none'.
  options: (bool) True,False
  default: False
  WARNING: Currently this just asks whether you accept
  the displayed calibration and if not, continues
  without doing any calibration.

outfile -- Name of output file
  default: '' (<sdfile>_cal)

outform -- format of output file
  options: 'ASCII', 'SDFITS', 'MS', 'ASAP'
  default: 'ASAP'

```

```

    example: the ASAP format is easiest for further sd
              processing; use MS for CASA imaging.
              If ASCII, it then will append some stuff to
              the outfile name
overwrite -- overwrite the output file if it already exists
options: (bool) True,False
default: False
WARNING: if outform='ASCII', this parameter is ignored
plotlevel -- control for plotting of results
options: (int) 0=none, 1=some, 2=more, <0=hardcopy
default: 0 (no plotting)
example: plotlevel<0 as abs(plotlevel), e.g.
        -1 => hardcopy of final plot (will be named
        <outfile>_calspec.eps)
WARNING: be careful plotting in fsotf mode!

```

DESCRIPTION:

Task `sdaverage` performs data selection, calibration for single-dish spectra. By setting `calmode='none'` one can run `sdaverage` on already calibrated data, for further selection, averaging and atmospheric optical depth correction.

If you give multiple IFs in `iflist`, then your scantable will have multiple IFs. This can be handled, but there can be funny interactions later on. We recommend you split each IF out into separate files by re-running `sdaverage` with each IF in turn unless you want to averaging of multi-resolution spectra (see below).

To save the output spectra in a certain range of channels, you set the range in `channelrange`. Averaging of multi-resolution spectra can be achieved by setting the sub-parameter in `timeaverage`, `averageall` to True. It generally handles multi-IFs by selecting overlaps in IFs and assigning new IFs in the output spectra.

ASAP recognizes the data of the "AT" telescopes, but currently does not know about the GBT or any other telescope. This task does know about GBT. Telescope name is obtained from the data. If you wish to change the `fluxunit` (see below) by unsetting the sub-parameter `unset` (`telescopeparm=''`), it will use internal telescope parameters for flux conversion for the data from AT telescopes and it will use an approximate aperture efficiency conversion for the GBT data. When `telescopeparm` is a list, then if the list has a single float it is assumed to be the gain in Jy/K. If there are two or more elements they are assumed to be telescope diameter (m) and aperture efficiency respectively.

`sdaverage` assumes that `fluxunit` is set correctly in the data already. If not, then set `telescopeparm='FIX'` and it will set the default units to `fluxunit` without conversion. NOTE: If the data table in `sdfile` is an ms from GBT and the default flux unit is missing, this task automatically fixes the default `fluxunit` to 'K' before performing the conversion.

Notes:

- If the observation mode is On-The-Fly, the direction information will be considered for averaging, i.e., spectra will be averaged only if the directions are in agreement with each other.
- Calibration of frequency switching data is now fully implemented, including folding.
- Additional calibration algorithms are implemented. These are the 'Chopper-Wheel' calibration and the one adopted in APEX telescope (an advanced version of classical 'Chopper-Wheel' method). Two load calibration, which will be a standard calibration mode of ALMA total power antennas, is formally implemented but still experimental.

A.2.1.2 sdsMOOTH

Keyword arguments:

```

sdfilename -- name of input SD dataset
antenna -- antenna name or id (only effective for MS input)
scanaverage -- average integrations within scans
    options: (bool) True,False
    default: False
    example: if True, this happens in read-in
             For GBT, set False!
scanlist -- list of scan numbers to process
    default: [] (use all scans)
    example: [21,22,23,24]
             this selection is in addition to field,
             iflist, and pollist
field -- selection string for selecting scans by name
    default: '' (no name selection)
    example: 'FLS3a*'
             this selection is in addition to scanlist,
             iflist,pollist
iflist -- list of IF id numbers to select
    default: [] (use all IFs)
    example: [15]
             this selection is in addition to scanlist,
             field, and pollist
pollist -- list of polarization id numbers to select
    default: [] (use all polarizations)
    example: [1]
             this selection is in addition to scanlist,
             field, and iflist
kernel -- type of spectral smoothing
    options: 'hanning','gaussian','boxcar'
    default: 'hanning'

```

>>>kernel expandable parameter

```

    kwidth -- width of spectral smoothing kernel
            options: (int) in channels
            default: 5
            example: 5 or 10 seem to be popular for boxcar,
                    ignored for hanning (fixed at 5 chans)
                    (0 will turn off gaussian or boxcar)

verify -- verify the results of smoothing
        options: (bool) True,False
        default: False
        WARNING: Currently this just asks whether you accept
                the displayed smoothing and if not, continues
                without smoothing.

outfile -- Name of output ASAP format(scantable) file
         default: '' (<sdfile>_sm)

outform -- format of output file
         options: 'ASCII','SDFITS','MS','ASAP'
         default: 'ASAP'
         example: the ASAP format is easiest for further sd
                 processing; use MS for CASA imaging.
                 If ASCII, then will append some stuff to
                 the outfile name

overwrite -- overwrite the output file if already exists
           options: (bool) True,False
           default: False
           WARNING: if outform='ASCII', this parameter is ignored

plotlevel -- control for plotting of results
           options: (int) 0=none, 1=some, 2=more, <0=hardcopy
           default: 0 (no plotting)
           example: plotlevel<0 as abs(plotlevel), e.g.
                   -1 => hardcopy of final plot (will be named
                   <outfile>_smspec.eps)

```

DESCRIPTION:

Task `sdsMOOTH` performs smoothing of the single-dish spectra. Set `plotlevel <= 1` to plot the spectrum before and after smoothing.

A.2.1.3 sdbaseline**Keyword arguments:**

```

sdfile -- name of input SD dataset
antenna -- antenna name or id (only effective for MS input)
fluxunit -- units for line flux
           options: 'K','Jy',''

```

```

default: '' (keep current fluxunit)
WARNING: For GBT data, see description below

>>> fluxunit expandable parameter
telescopeparm -- the telescope characteristics
options: (str) name or (list) list of gain info
default: '' (none set)
example: if telescopeparm='', it tries to get the telescope
name from the data.
Full antenna parameters (diameter,ap.eff.) known
to ASAP are
'ATPKSMB', 'ATPKSHOH', 'ATMOPRA', 'DSS-43',
'CEDUNA', 'HOBART'. For GBT, it fixes default fluxunit
to 'K' first then converts to a new fluxunit.
telescopeparm=[104.9,0.43] diameter(m), ap.eff.
telescopeparm=[0.743] gain in Jy/K
telescopeparm='FIX' to change default fluxunit
see description below

specunit -- units for spectral axis
options: (str) 'channel','km/s','GHz','MHz','kHz','Hz'
default: '' (=current)
example: this will be the units for masklist
frame -- frequency frame for spectral axis
options: (str) 'LSRK','REST','TOPO','LSRD','BARY',
'GEO','GALACTO','LGROUP','CMB'
default: currently set frame in scantable
WARNING: frame='REST' not yet implemented
doppler -- doppler mode
options: (str) 'RADIO','OPTICAL','Z','BETA','GAMMA'
default: currently set to doppler in scantable
scanlist -- list of scan numbers to process
default: [] (use all scans)
example: [21,22,23,24]
this selection is in addition to field,
iflist, and pollist
field -- selection string for selecting scans by name
default: '' (no name selection)
example: 'FLS3a*'
this selection is in addition to scanlist,
iflist, and pollist
iflist -- list of IF id numbers to select
default: [] (use all IFs)
example: [15]
this selection is in addition to scanlist,

```

```

        field, and pollist
pollist -- list of polarization id numbers to select
        default: [] (use all polarizations)
        example: [1]
                this selection is in addition to scanlist,
                field, and iflist
tau -- atmospheric optical depth
        default: 0.0 (no correction)
blmode -- mode for baseline fitting
        options: (str) 'auto','list','interact'
        default: 'auto'
        example: blmode='auto' uses expandable parameters
                in addition to blpoly to run linefinder
                to determine line-free regions
                USE WITH CARE! May need to tweak the parameters,
                thresh, avg_limit, and edge.
                blmode='interact' allows adding and deleting mask
                regions by drawing rectangles on the plot with mouse.
                Draw a rectangle with LEFT-mouse to ADD the region to
                the mask and with RIGHT-mouse to DELETE the region.

>>> blmode expandable parameters
        thresh -- S/N threshold for linefinder
                default: 5
                example: a single channel S/N ratio above which the channel is
                        considered to be a detection
        avg_limit -- channel averaging for broad lines
                default: 4
                example: a number of consecutive channels not greater than
                        this parameter can be averaged to search for broad lines
        edge -- channels to drop at beginning and end of spectrum
                default: 0
                example: [1000] drops 1000 channels at beginning AND end
                        [1000,500] drops 1000 from beginning and 500 from end

Note: For bad baselines threshold should be increased,
and avg_limit decreased (or even switched off completely by
setting this parameter to 1) to avoid detecting baseline
undulations instead of real lines.

blpoly -- order of baseline polynomial
        options: (int) (<0 turns off baseline fitting)
        default: 5
        example: typically in range 2-9 (higher values
                seem to be needed for GBT)

```

```

verify -- verify the results of baseline fitting
  options: (bool) True,False
  default: False
  WARNING: Currently it just asks whether you accept
           the displayed fit and if not, continues
           without doing any baseline fit.
masklist -- list of mask regions to INCLUDE in BASELINE fit
  default: [] (entire spectrum)
  example: [[1000,3000],[5000,7000]]
           if blmode='auto' then this mask will be applied
           before fitting
outfile -- Name of output file
  default: '' (<sdfile>_bs)
outform -- format of output file
  options: 'ASCII','SDFITS','MS','ASAP'
  default: 'ASAP'
  example: the ASAP format is easiest for further sd
           processing; use MS for CASA imaging.
           If ASCII, then will append some stuff to
           the outfile name
overwrite -- overwrite the output file if already exists
  options: (bool) True,False
  default: False
  WARNING: if outform='ASCII', this parameter is ignored
plotlevel -- control for plotting of results
  options: (int) 0=none, 1=some, 2=more, <0=hardcopy
  default: 0 (no plotting)
  example: plotlevel<0 as abs(plotlevel), e.g.
           -1 => hardcopy of final plot (will be named
           <outfile>_bspec.eps)
  WARNING: be careful plotting in fsof mode!

```

DESCRIPTION:

Task `sdbaseline` performs baseline fitting/removal for single-dish spectra. The fit parameters, terms and rms of base-line are saved to an ASCII file, `<outfile>_blparam.txt`.

Also, see the notes on `fluxunit` and `telescopeparm` in the section for `sdaverage`. See the `sdaverage` description for information on `fluxunit` conversion and the `telescopeparm` parameter.

By setting `blmode='interact'`, you can set/unset mask regions interactively using mouse buttons. Current mask regions will be shown with yellow shading. Baseline fit parameters and rms of fitted spectra are automatically saved to an ASCII file, `<outfile>_blparam.txt`.

A.2.1.4 `sdcal`

Keyword arguments:

`sdfile` -- name of input SD dataset

`antenna` -- antenna name or id (only effective for MS input)

`fluxunit` -- units for line flux

options: 'K','Jy',''

default: '' (keep current fluxunit)

WARNING: For GBT data, see description below.

>>> `fluxunit` expandable parameter

`telescopeparm` -- the telescope characteristics

options: (str) name or (list) list of gain info

default: '' (none set)

example: if `telescopeparm`='', it tries to get the telescope name from the data.

Full antenna parameters (diameter,ap.eff.) known to ASAP are

'ATPKSMB', 'ATPKSHOH', 'ATMOPRA', 'DSS-43',

'CEDUNA', 'HOBART'. For GBT, it fixes default fluxunit

to 'K' first then converts to a new fluxunit.

`telescopeparm`=[104.9,0.43] diameter(m), ap.eff.

`telescopeparm`=[0.743] gain in Jy/K

`telescopeparm`='FIX' to change default fluxunit

see description below

`specunit` -- units for spectral axis

options: (str) 'channel','km/s','GHz','MHz','kHz','Hz',''

default: '' (=current)

example: this will be the unit for masklist

`frame` -- frequency frame for spectral axis

options: (str) 'LSRK','REST','TOPO','LSRD','BARY',

'GEO','GALACTO','LGROUP','CMB'

default: currently set frame in scantable

WARNING: `frame`='REST' not yet implemented

`doppler` -- doppler mode

options: (str) 'RADIO','OPTICAL','Z','BETA','GAMMA'

default: currently set doppler in scantable

`calmode` -- calibration mode

options: 'ps','nod','fs','fsotf','quotient','none'

default: 'none'

example: choose mode 'none' if you have already calibrated and want to try baselines or averaging

`scanlist` -- list of scan numbers to process

```

    default: [] (use all scans)
    example: [21,22,23,24]
           this selection is in addition to field,
           iflist, and pollist
field -- selection string for selecting scans by name
    default: '' (no name selection)
    example: 'FLS3a*'
           this selection is in addition to scanlist,
           iflist, and pollist
iflist -- list of IF id numbers to select
    default: [] (use all IFs)
    example: [15]
           this selection is in addition to scanlist,
           field, and pollist
pollist -- list of polarization id numbers to select
    default: [] (use all polarizations)
    example: [1]
           this selection is in addition to scanlist,
           field, and iflist
channelrange -- channel range selection
    default: [] (use all channel)
    example: [0,5000]
           Note that specified values are recognized as
           'channel' regardless of the value of specunit
average -- averaging on spectral data
    options: (bool) True,False
    default: False

>>> average expandable parameter
    scanaverage -- average integrations within scans
        options: (bool) True,False
        default: False
        example: if True, this happens in read-in
                 For GBT, set False!
    timeaverage -- average times for multiple scan cycles
        options: (bool) True,False
        default: False
        example: if True, this happens after calibration
    tweight -- weighting for time average
        options: 'none'
                'var' (1/var(spec) weighted)
                'tsys' (1/Tsys**2 weighted)
                'tint' (integration time weighted)
                'tintsys' (Tint/Tsys**2)
                'median' (median averaging)

```

```

        default: 'none'
averageall -- average multi-resolution spectra
             spectra are averaged by referring
             their frequency coverage
        default: False
polaverage -- average polarizations
             options: (bool) True,False
             default: False
pweight -- weighting for polarization average
           options: 'none'
                'var' (1/var(spec) weighted)
                'tsys' (1/Tsys**2 weighted)

tau -- atmospheric optical depth
     default: 0.0 (no correction)
kernel -- type of spectral smoothing
         options: 'none', 'hanning', 'gaussian', 'boxcar'
         default: 'none'

>>> kernel expandable parameter
     kwidth -- width of spectral smoothing kernel
             options: (int) in channels
             default: 5
             example: 5 or 10 seems to be popular for boxcar
                    ignored for hanning (fixed at 5 chans)
                    (0 will turn off gaussian or boxcar)

blmode -- mode for baseline fitting
         options: (str) 'none', 'auto', 'list', 'interact'
         default: 'none'
         example: blmode='auto' uses expandable parameters
                in addition to blpoly to run linefinder
                to determine line-free regions
                USE WITH CARE! May need to tweak the parameters,
                thresh, avg_limit, and edge.
                blmode='interact' allows adding and deleting mask
                regions by drawing rectangles on the plot with mouse.
                Draw a rectangle with LEFT-mouse to ADD the region to
                the mask and with RIGHT-mouse to DELETE the region.

>>> blmode expandable parameters
     thresh -- S/N threshold for linefinder
             default: 5
             example: a single channel S/N ratio above which the channel is
                    considered to be a detection

```

```

avg_limit -- channel averaging for broad lines
            default: 4
            example: a number of consecutive channels not greater than
                    this parameter can be averaged to search for broad lines
edge -- channels to drop at beginning and end of spectrum
        default: 0
        example: [1000] drops 1000 channels at beginning AND end
                [1000,500] drops 1000 from beginning and 500 from end

```

Note: For bad baselines threshold should be increased, and avg_limit decreased (or even switched off completely by setting this parameter to 1) to avoid detecting baseline undulations instead of real lines.

```

blpoly -- order of baseline polynomial
         options: (int) (<0 turns off baseline fitting)
         default: 5
         example: typically in range 2-9 (higher values
                 seem to be needed for GBT)
verifical -- verify the results of calibration
            options: (bool) True,False
            default: False
            WARNING: Currently verifying parameters just asks whether you
                    accept the displayed calibraion/fit and if not,
                    continues without doing any calibraion/baseline fit.
verifysm -- verify the results of smoothing
            options: (bool) True,False
            default: False
verifybl -- verify the results of baseline fitting
            options: (bool) True,False
            default: False
masklist -- list of mask regions to INCLUDE in BASELINE fit
            default: [] (entire spectrum)
            example: [[1000,3000],[5000,7000]]
                   if blmode='auto' then this mask will be applied
                   before fitting
outfile -- Name of output file
          default: '' (<sdfile>_cal)
outform -- format of output file
          options: 'ASCII','SDFITS','MS','ASAP'
          default: 'ASAP'
          example: the ASAP format is easiest for further sd
                  processing; use MS for CASA imaging.
                  If ASCII, then will append some stuff to
                  the outfile name

```

```

overwrite -- overwrite the output file if already exists
  options: (bool) True,False
  default: False
  WARNING: if outform='ASCII', this parameter is ignored
plotlevel -- control for plotting of results
  options: (int) 0=none, 1=some, 2=more, <0=hardcopy
  default: 0 (no plotting)
  example: plotlevel<0 as abs(plotlevel), e.g.
           -1 => hardcopy of final plot (will be named
           <outfile>_calspec.eps)
  WARNING: be careful plotting in fsotf mode!

```

DESCRIPTION:

Task `sdcal` performs data selection, calibration, and/or spectral baseline fitting for single-dish spectra. This task internally calls the tasks `sdaverage`, `sdsMOOTH`, and `sdbaseline`, and it can be used to run all the three steps in one task execution. By setting `calmode='none'` one can run `sdcal` on already calibrated data for further selection, averaging and atmospheric optical depth correction. See the `sdaverage` description for information on the `fluxunit` conversion and the `telescopeparm` parameter. To save the output spectra within a certain range of channels, you set the range in `channelrange`. Averaging of multi-resolution spectra can be achieved by setting the sub-parameter of `average`, `averageall`, to `True`. It generally handles multi-IFs by selecting overlaps in IFs and assigning new IFs in the output spectra.

The `verifycal`, `verifysm`, and `verifybl` parameters correspond to parameter `verify` in `sdaverage`, `sdsMOOTH`, and `sdbaseline`, respectively.

A.2.1.5 `sdcoadd`

Keyword arguments:

```

sdfilelist -- list of names of input SD dataset
antenna -- antenna name or id (only effective for MS input)
fluxunit -- units for line flux
  options: 'K','Jy',''
  default: '' (keep current fluxunit)
  WARNING: For GBT data, see description below.

```

```

>>> fluxunit expandable parameter
  telescopeparm -- the telescope characteristics
    options: (str) name or (list) list of gain info
    default: '' (none set)
    example: if telescopeparm='', it tries to get the telescope
             name from the data.
             Full antenna parameters (diameter,ap.eff.) known
             to ASAP are

```

'ATPKSMB', 'ATPKSHOH', 'ATMOPRA', 'DSS-43',
 'CEDUNA', 'HOBART'. For GBT, it fixes default fluxunit
 to 'K' first then convert to a new fluxunit.
 telescopeparm=[104.9,0.43] diameter(m), ap.eff.
 telescopeparm=[0.743] gain in Jy/K
 telescopeparm='FIX' to change default fluxunit
 see description below

specunit -- units for spectral axis
 options: (str) 'channel', 'km/s', 'GHz', 'MHz', 'kHz', 'Hz'
 default: '' (=current)
 example: this will be the units for masklist

frame -- frequency frame for spectral axis
 options: (str) 'LSRK', 'REST', 'TOPO', 'LSRD', 'BARY',
 'GEO', 'GALACTO', 'LGROUP', 'CMB'
 default: currently set frame in scantable
 WARNING: frame='REST' not yet implemented

doppler -- doppler mode
 options: (str) 'RADIO', 'OPTICAL', 'Z', 'BETA', 'GAMMA'
 default: currently sets doppler in scantable

scanaverage -- average integrations within scans
 options: (bool) True,False
 default: False
 example: if True, this happens in read-in
 For GBT, set False!

timeaverage -- average times for multiple scan cycles
 options: (bool) True,False
 default: False
 example: if True, this happens after calibration

polaverage -- average polarizations
 options: (bool) True,False
 default: False

outfile -- Name of output file
 default: '' (scantable)
 example:

outform -- format of output file
 options: 'ASCII', 'SDFITS', 'MS', 'ASAP'
 default: 'ASAP'
 example: the ASAP format is easiest for further sd
 processing; use MS for CASA imaging.
 If ASCII, then will append some stuff to
 the outfile name

overwrite -- overwrite the output file if it already exists
 options: (bool) True,False
 default: False

WARNING: if outform='ASCII', this parameter is ignored

DESCRIPTION:

Task `sdcoadd` merges multiple single dish spectral data given by a list of spectral data file names in any of the following formats, ASAP, MS2, and SDFITS. The units of line flux, the units of spectral axis, frame, and doppler are assumed to be those of the first one in the `sdfilelist` if not specified. The `timaverager` and `polaverager` are used to perform time and polarization averaging over scans on the merged scantable to obtained co-added spectra before saving to a file on disk.

A.2.1.6 `sdflag`

Keyword arguments:

`sdfile` -- name of input SD dataset
`antenna` -- antenna name or id (only effective for MS input)
`scanlist` -- list of scan numbers to process
 default: [] (use all scans)
 example: [21,22,23,24]
 this selection is in addition to field
 and iflist
`field` -- selection string for selecting scans by name
 default: '' (no name selection)
 example: 'FLS3a*'
 this selection is in addition to scanlist
 and iflist
`iflist` -- list of IF id numbers to select
 default: [] (use all IFs)
 example: [15]
 this selection is in addition to scanlist
 and field
`pollist` -- list of polarization id numbers to select
 default: [] (use all polarizations)
 example: [1]
 this selection is in addition to scanlist,
 field, and iflist
`maskflag` -- list of mask regions to apply flag/unflag
 default: [] (entire spectrum)
 example: [[1000,3000],[5000,7000]]
 WARNING: if one or more rows are given in flagrow, or clip=True,
 this parameter is ignored
`flagrow` -- list of row numbers to apply flag/unflag (row-based)
 default: [] (no row selection)
 example: [0, 2, 3]
 WARNING: this parameter is effective only when
 one or more row numbers are given explicitly and also clip=False

```

clip -- flag data that are outside a specified range
      options: (bool) True, False
      default: False
  >>> clip expandable parameters
      clipminmax -- range of data that will NOT be flagged
                  default: [] means do not use clip option
                  example: [0.0,1.5]
      clipoutside -- clip OUTSIDE the range ?
                  options: (bool) True, False
                  default: True
                  example: clipoutside=False means flag data WITHIN the range.

flagmode -- flag mode
          default: 'flag'
          options: 'flag','unflag','restore'
                  in 'restore' mode, a history of flagging is
                  displayed and current flag state is returned

outfile -- Name of output file
          default: '' (<sdfile>_f)

outform -- format of output file
          options: 'ASCII','SDFITS','MS','ASAP'
          default: 'ASAP'
          example: the ASAP format is easiest for further sd
                  processing; use MS for CASA imaging.

overwrite -- overwrite the output file if it already exists
            options: (bool) True,False
            default: False
            WARNING: if outform='ASCII', this parameter is ignored

plotlevel -- control for plotting of results
            options: (int) 0=none, 1=some, 2=more, <0=hardcopy
            default: 0 (no plotting)
            example: plotlevel<0 as abs(plotlevel), e.g.
                    -1 => hardcopy of final plot (will be named
                    <outfile>_flag.eps)
            WARNING: be careful plotting in fsotf mode!

```

Returns: Current values of maskflag for each spectrum as a list,
only if flagmode is set to 'restore'. Note that the
'restore' mode refers to scanlist, iflist, and pollist.

DESCRIPTION:

Task `sdflag` performs simple channel/row based flagging on spectra. Currently, the available ways of flagging include: (1) channel based flagging by specifying regions in channel, (2) channel based

flagging by specifying a range of spectral value, and (3) row based flagging by specifying a list of row numbers. These three ways of flagging can not be executed simultaneously. The first one (channel based) can be executed by giving flag regions in `maskflag`, no row in `flagrow`, and setting `clip=False`. The second one (channel based intensity clipping) can run when you set `clip=True` and specify a range in `clipminmax` (the values given in `maskflag` and `flagrow` are ignored). The third one (row based flagging) is executed when one or more rows are specified in `flagrow` and `clip` is set `False`. This is not interactive flagging. If `plotlevel>=1`, the task asks you if you really apply the flags before it is actually written to the data with a plot indicating flagged regions. The flags are not written to the current (input) datasets unless `outfile='none'`. Flagged regions (or values of flag masks already applied) of the input data can be listed by setting `flagmode='restore'`. It will print out a summary on the screen and returns `maskflag` values for each spectrum as a list. Please note that this task is still experimental.

A.2.1.7 `sdfit`

Keyword arguments:

```

sdfit -- name of input SD dataset
        default: none - must input file name
        example: 'mysd.asap'
           See sdcal for allowed formats.
antenna -- antenna name or id (only effective for MS input)
fluxunit -- units for line flux
            options: (str) 'K','Jy',''
            default: '' (keep current fluxunit)
            WARNING: For GBT data, see description below.

>>> fluxunit expandable parameter
        telescopeparm -- the telescope characteristics
            options: (str) name or (list) list of gain info
            default: '' (none set)
            example: if telescopeparm='', it tries to get the telescope
                    name from the data.
                    Full antenna parameters (diameter,ap.eff.) known
                    to ASAP are
                    'ATPKSMB', 'ATPKSHOH', 'ATMOPRA', 'DSS-43',
                    'CEDUNA','HOBART'. For GBT, it fixes default fluxunit
                    to 'K' first then converts to a new fluxunit.
                    telescopeparm=[104.9,0.43] diameter(m), ap.eff.
                    telescopeparm=[0.743] gain in Jy/K
                    telescopeparm='FIX' to change default fluxunit
                    see description below

specunit -- units for spectral axis
            options: (str) 'channel','km/s','GHZ','MHZ','kHz','Hz',''
```

```

    default: '' (=current)
frame -- frequency frame for spectral axis
    options: (str) 'LSRK','REST','TOPO','LSRD','BARY',
               'GEO','GALACTO','LGROUP','CMB'
    default: currently set frame in scantable
    WARNING: frame='REST' not yet implemented
doppler -- doppler mode
    options: (str) 'RADIO','OPTICAL','Z','BETA','GAMMA'
    default: currently set doppler in scantable
scanlist -- list of scan numbers to process
    default: [] (use all scans)
    example: [21,22,23,24]
field -- selection string for selecting scans by name
    default: '' (no name selection)
    example: 'FLS3a*'
           this selection is in addition to scanlist
           and iflist
iflist -- list of IF id numbers to select
    default: [] (use all IFs)
    example: [15]
pollist -- list of polarization id numbers to select
    default: [] (use all polarizations)
    example: [1]
fitfunc -- function for fitting
    options: (str) 'gauss','lorentz'
    default: 'gauss'
fitmode -- mode for fitting
    options: (str) 'list','auto','interact'
    default: 'auto'
    example: 'list' will use maskline to define regions to
           fit for lines with nfit in each
           'auto' will use the linefinder to fit for lines
           using the following parameters
           'interact' allows adding and deleting mask
           regions by drawing rectangles on the plot
           with the mouse. Draw a rectangle with LEFT-mouse
           to ADD the region to the mask and with RIGHT-mouse
           to DELETE the region.

>>> fitmode expandable parameters
    thresh -- S/N threshold for linefinder
           default: 5
           example: a single channel S/N ratio above which the channel
                   is considered to be a detection
    min_nchan -- minimum number of consecutive channels for linefinder

```

```

        default: 3
        example: minimum number of consecutive channels required to
                pass threshold
    avg_limit -- channel averaging for broad lines
        default: 4
        example: a number of consecutive channels not greater than
                this parameter can be averaged to search for broad
                lines
    box_size -- running mean box size
        default: 0.2
        example: a running mean box size specified as a fraction
                of the total spectrum length
    edge -- channels to drop at beginning and end of spectrum
        default: 0
        example: [1000] drops 1000 channels at beginning AND end
                [1000,500] drops 1000 from beginning and 500 from
                end

        Note: For bad baselines thresh should be increased,
        and avg_limit decreased (or even switched off completely by
        setting this parameter to 1) to avoid detecting baseline
        undulations instead of real lines.

maskline -- list of mask regions to INCLUDE in LINE fitting
    default: all
    example: maskline=[[3900,4300]] for a single region, or
            maskline=[[3900,4300],[5000,5400]] for two, etc.
invertmask -- invert mask (EXCLUDE masklist instead)
    options: (bool) True, False
    default: False
    example: invertmask=True, then will make one region that is
            the exclusion of the maskline regions
nfit -- list of number of gaussian lines to fit in in maskline region
    default: 0 (no fitting)
    example: nfit=[1] for single line in single region,
            nfit=[2] for two lines in single region,
            nfit=[1,1] for single lines in each of two regions, etc.
fitfile -- name of output file for fit results
    default: no output fit file
    example: 'mysd.fit'
overwrite -- overwrite the fitfile if already exists
    options: (bool) True, False
    default: False
plotlevel -- control for plotting of results
    options: (int) 0=none, 1=some, 2=more

```

```

default: 0 (no plotting)
example: plotlevel=1 plots fit
         plotlevel=2 plots fit and residual
         no hardcopy available for fitter
WARNING: be careful plotting OTF data with lots of fields

```

```

-----
Returns a Python dictionary of line statistics
keys:    'peak','cent','fwhm','nfit'
example: each value is a list of lists with one list of
         2 entries [fitvalue,error] per component.
         e.g. xstat['peak']=[[234.9, 4.8],[234.2, 5.3]]
         for 2 components.

```

DESCRIPTION:

Task `sdfit` is a basic line-fitter for single-dish spectra. It assumes that the spectra have been calibrated in `sdaverage` or `sdcal`.

Furthermore, it assumes that any selection of scans, IFs, polarizations, and time and channel averaging/smoothing has also already been done (in other `sd` tasks) as there are no controls for these. Note that you can use `sdsave` to do selection and write out a new scantable.

Note that multiple scans and IFs can in principle be handled, but we recommend that you use `scanlist`, `field`, and `iflist` to give a single selection for each fit.

Currently you can choose Gaussian or Lorentzian profile as a fitting model.

Interactive mask selection for spectral line fitting is enabled with `fitmode='interact'`.

For complicated spectra, `sdfit` does not do a good job of "auto-guessing" the starting model for the fit. We recommend you use `sd.fitter` in the toolkit which has more options, such as fixing components in the fit and supplying starting guesses by hand.

See the `sdaverage` description for information on `fluxunit` conversion and the `telescopeparm` parameter.

A.2.1.8 `sdimaging`

Keyword arguments:

```

sdfile -- name of input SD (MS) dataset
specunit -- units for spectral axis
          options: (str) 'channel','km/s','GHz','MHz','kHz','Hz'
          default: 'channel'
          example: these will be the units for nchan, start, and step

```

```
>>> specunit='km/s' expandable parameters
```

```

restfreq -- rest frequency
           default: '' (refer input data)
           example: 1.0e11, '100GHz'

scanlist -- list of scan numbers to process
           default: [] (use all scans)
           example: [21,22,23,24]
                this selection is in addition to field and spw
field -- field id or selection string for selecting scans by name
        default: 0
        example: 'FLS3a', 0
                this selection is in addition to scanlist and spw
spw -- spectral window id
       default: 0
       example: 1
                this selection is in addition to scanlist and field
antenna -- select data based on antenna name(s) or id(s)
          default: 0
          example: '0', 'DV01'
stokes -- select data based on stokes or polarization type
         default: '' (use all polarizations)
         example: 'XX'
gridfunction -- gridding function for imaging
              options: 'BOX' (Box-car), 'SF' (Spheroidal), 'PB' (Primary-beam)
              default: 'BOX'
              example: 'SF'
imagename -- output image name
           default: none
           example: 'mySDimage.im'
overwrite -- overwrite option for imagename
           default: False (not overwrite)
           options: True, False
           example: if True, existing file will be overwritten
imsize -- x and y image size in pixels, symmetric for single value
         default: [256,256]
         example: imsize=200 (equivalent to [200,200])
cell -- x and y cell size. default unit arcmin
       default: ['1.0arcmin', '1.0arcmin']
       example: cell=['0.2arcmin', '0.2arcmin']
                cell='0.2arcmin' (equivalent to example above)
dochannelmap -- channel map image or total power image
              default: False (total power)
              options: True (channel map), False

>>> dochannelmap=True expandable parameters

```

```

nchan -- number of spectral channel for created image
        default: 1
        example: to do total power imaging, set -1
start -- reference value of start channel (in units of specunit)
        default: 0 (0th channel if specunit='channel')
        example: 100
step -- width of each spectral channel for created image
        default: 1 (channel width of 1 channel if specunit='channel')
        aexample: 100

phasecenter -- image phase center: direction measure or fieldid
        default: 0
        example: 'J2000 13h44m00 -17d02m00', 'AZEL -123d48m29 15d41m41'
ephemsrcname -- ephemeris source name for moving source
        default: ''
        if the source name in the data matches one of the known
        solar objects by the system, this task automatically set
        the source name.
        example: 'moon'
pointingcolumn -- pointing data column to use
        option: 'direction', 'target', 'pointing_offset', 'source_offset', 'encoder'
        default: 'direction'

```

DESCRIPTION:

Task `sdimaging` creates an image from input single-dish data. The input can be either total power or spectral data. Currently, this task directly accesses the Measurement Set data only because of the data access efficiently. It differs from other single-dish tasks that mostly operate on the ASAP scantable data format.

Units of spectral axis can be specified via a parameter `specunit`. Allowed values for `specunit` are 'channel', 'GHz', 'MHz', 'kHz', 'Hz', and 'km/s'. This parameter is also used as the units of the parameter `start` and `step` that specify reference value of start channel and width of each spectral channel for channel map, respectively.

Selection of the antennas can be made by setting `antennaid(s)` or `antenna name(s)` in string (e.g. '0', 'DV01', etc.). Note that the task doesn't work for multiple antenna selection and/or multiple field selection.

The parameter `gridfunction` sets gridding function for imaging. Currently, the task supports 'BOX' (Box-car), 'SF' (Prolate Spheroidal Wave Function), and 'PB' (Primary Beam). For 'PB', correct antenna information should be included in input file.

A.2.1.9 sdlist

Keyword arguments:

```

sdfile -- name of input SD dataset
antenna -- antenna name or id (only effective for MS input)
scanaverage -- average integrations within scans
    options: (bool) True,False
    default: False
    example: if True, this happens in read-in
             For GBT, set False!
listfile -- Name of output file for summary list
    default: '' (no output file)
    example: 'mysd_summary.txt'
overwrite -- overwrite the output file if already exists
    options: (bool) True,False
    default: False

```

DESCRIPTION:

Task `sdlist` lists the scan summary of the dataset after importing as a scantable into ASAP. It will optionally output this summary as file.

Note that if your `PAGER` environment variable is set to 'less' and the 'verbose' ASAP environment variable to True (the default), then the screen version of the summary will page. You can disable this for `sdlist` by setting `sd.rcParams['verbose']=False` before running `sdlist`. Set it back afterward if you want lots of information.

A.2.1.10 sdmath

Keyword arguments:

```

expr -- Mathematical expression using scantables
varlist -- Dictionary of variables in expr and their values.
    Keys must coincide with variables used in expr.
    Values are substituted in each value in expr.
antenna -- antenna name or id (only effective for MS input).
fluxunit -- units for line flux
    options: 'K','Jy',''
    default: '' (keep current fluxunit)
WARNING: For GBT data, see description below.

```

```

>>> fluxunit expandable parameter
    telescopeparm -- the telescope characteristics
        options: (str) name or (list) list of gain info
        default: '' (none set)
        example: if telescopeparm='', it tries to get the telescope
                 name from the data.
                 Full antenna parameters (diameter,ap.eff.) known
                 to ASAP are

```

'ATPKSMB', 'ATPKSHOH', 'ATMOPRA', 'DSS-43',
 'CEDUNA', 'HOBART'. For GBT, it fixes default fluxunit
 to 'K' first then converts to a new fluxunit.
 telescopeparm=[104.9,0.43] diameter(m), ap.eff.
 telescopeparm=[0.743] gain in Jy/K
 telescopeparm='FIX' to change default fluxunit
 see description below

specunit -- units for spectral axis
 options: (str) 'channel', 'km/s', 'GHz', 'MHz', 'kHz', 'Hz'
 default: '' (=current)
 example: these will be the units for masklist

frame -- frequency frame for spectral axis
 options: (str) 'LSRK', 'REST', 'TOPO', 'LSRD', 'BARY',
 'GEO', 'GALACTO', 'LGROUP', 'CMB'
 default: currently set frame in scantable
 WARNING: frame='REST' not yet implemented

doppler -- doppler mode
 options: (str) 'RADIO', 'OPTICAL', 'Z', 'BETA', 'GAMMA'
 default: currently set doppler in scantable

scanlist -- list of scan numbers to process
 default: [] (use all scans)
 example: [21,22,23,24]
 this selection is in addition to field,
 iflist, and pollist

field -- selection string for selecting scans by name
 default: '' (no name selection)
 example: 'FLS3a*'
 this selection is in addition to scanlist,
 iflist, and pollist

iflist -- list of IF id numbers to select
 default: [] (use all IFs)
 example: [15]
 this selection is in addition to scanlist,
 field, and pollist

pollist -- list of polarization id numbers to select
 default: [] (use all polarizations)
 example: [1]
 this selection is in addition to scanlist,
 field, and iflist

outfile -- Name of output file
 default: '' (<sdfile>_cal)

outform -- format of output file
 options: 'ASCII', 'SDFITS', 'MS', 'ASAP'
 default: 'ASAP'


```

example: the ASAP format is easiest for further sd
         processing; use MS for CASA imaging.
         If ASCII, then will append some stuff to
         the outfile name
overwrite -- overwrite the output file if already exists
options: (bool) True,False
default: False
WARNING: if outform='ASCII', this parameter is ignored

```

DESCRIPTION:

Task `sdmath` executes a mathematical expression for single dish spectra. The spectral data file can be any of the formats supported by ASAP (scantable, MS, rpfits, and SDFITS). In the expression, these file names should be put inside of single or double quotes. You can use variables in the expression. If you want to use, you must define `varlist` dictionary. Name of variables should be simple, e.g. V0, V1, etc., to avoid unexpected error. Keys of `varlist` must be name of variables that you used in the expression, and their values will be substituted for variables in the expression. Allowed type for value is numerical values, one- or two-dimensional lists (either Python list or `numpy.ndarray`), and filename strings that indicate spectral data or ASCII text, which is space-separated list of numerical values consisting of adequate number of rows and columns.

The `fluxunit`, `specunit`, and `frame` can be set, otherwise, the current settings of the first spectral data in the expression are used. Other selections (e.g. scan No, . IF, Pol) also apply to all the spectral data in the expression, so if any of the data are not selected, the task will produce no output.

Example:

```

# do on-off/off calculation
expr='("orion_on_data.asap"-orion_off_data.asap)/orion_off_data.asap'
outfile='orion_cal.asap'
sdmath()

# do on-off/off calculation using varlist
expr='V0/V1-V2'
varlist['V0']='orion_on_data.asap'
varlist['V1']='orion_off_data.asap'
varlist['V2']=1.0
outfile='orion_cal.asap'
sdmath()

# interpretation of ASCII file value for varlist
If the contents of input ASCII file is shown as,

0.5 0.3 0.2
1.0 0.2 0.9

```

it is interpreted as a list, `[[0.5,0.3,0.2],[1.0,0.2,0.9]]`.

A.2.1.11 `sdplot`

Keyword arguments:

`sdfilename` -- name of input SD dataset

`antenna` -- antenna name or id (only effective for MS input).

`fluxunit` -- units for line flux

options: 'K','Jy',''

default: '' (keep current fluxunit)

WARNING: For GBT data, see description below.

>>> `fluxunit` expandable parameter

`telescopeparm` -- the telescope characteristics

options: (str) name or (list) list of gain info

default: '' (none set)

example: if `telescopeparm=''`, it tries to get the telescope name from the data.

Full antenna parameters (diameter,ap.eff.) known to ASAP are

'ATPKSMB', 'ATPKSHOH', 'ATMOPRA', 'DSS-43',

'CEDUNA','HOBART'. For GBT, it fixes default fluxunit to 'K' first then converts to a new fluxunit.

`telescopeparm=[104.9,0.43]` diameter(m), ap.eff.

`telescopeparm=[0.743]` gain in Jy/K

`telescopeparm='FIX'` to change default fluxunit

see description below

`specunit` -- units for spectral axis

options: (str) 'channel','km/s','GHz','MHz','kHz','Hz'

default: '' (=current)

example: these will be the units for masklist

`restfreq` -- rest frequency used for `specunit='km/s'`

default: '' (use current setting)

example: 4.6e10 (float value), '46GHz' (string with unit)

Allowed units are 'THz', 'GHz', 'MHz', 'kHz', and 'Hz'

`frame` -- frequency frame for spectral axis

options: (str) 'LSRK','REST','TOPO','LSRD','BARY',
'GEO','GALACTO','LGROUP','CMB'

default: set current frame in scantable

WARNING: frame='REST' not yet implemented

`doppler` -- doppler mode

options: (str) 'RADIO','OPTICAL','Z','BETA','GAMMA'

default: set current doppler in scantable

```

scanlist -- list of scan numbers to process
  default: [] (use all scans)
  example: [21,22,23,24]
           this selection is in addition to field
           iflist and pollist
field -- selection string for selecting scans by name
  default: '' (no name selection)
  example: 'FLS3a*'
           this selection is in addition to scanlist
           iflist and pollist
iflist -- list of IF id numbers to select
  default: [] (use all IFs)
  example: [15]
           this selection is in addition to scanlist
           field and pollist
pollist -- list of polarization id numbers to select
  default: [] (use all polarizations)
  example: [1]
           this selection is in addition to scanlist,
           field, and iflist
scanaverage -- average ints within scans
  options: (bool) True,False
  default: False
timeaverage -- average times for multiple scan cycles
  options: (bool) True,False
  default: False
  example: if True, this happens after calibration

```

```

>>>timeaverage expandable parameter
  tweight -- weighting for time average
           options: 'none'
                'var'   (1/var(spec) weighted)
                'tsys'  (1/Tsys**2 weighted)
                'tint'  (integration time weighted)
                'tintsys' (Tint/Tsys**2)
                'median' ( median averaging)
           default: 'none'

```

```

polaverage -- average polarizations
  options: (bool) True,False
  default: False

```

```

>>>polaverage expandable parameter
  pweight -- weighting for polarization average
           options: 'none'

```

```

        'var' (1/var(spec) weighted)
        'tsys' (1/Tsys**2 weighted)

kernel -- type of spectral smoothing
  options: 'hanning','gaussian','boxcar', 'none'
  default: 'none'

>>>kernel expandable parameter
  kwidth -- width of spectral smoothing kernel
    options: (int) in channels
    default: 5
    example: 5 or 10 seem to be popular for boxcar
             ignored for hanning (fixed at 5 chans)
             (0 will turn off gaussian or boxcar)

plottype -- type of plot
  options: 'spectra','totalpower','pointing','azel'
  default: 'spectra'
stack -- code for stacking on single plot for spectral plotting
  options: 'p','b','i','t','s' or
           'pol','beam','if','time','scan'
  default: 'p'
  example: maximum of 25 stacked spectra
           stack by pol, beam, if, time, scan
panel -- code for splitting into multiple panels for spectral plotting
  options: 'p','b','i','t','s' or
           'pol','beam','if','time','scan'
  default: 'i'
  example: maximum of 25 panels
           panel by pol, beam, if, time, scan
flrange -- range for flux axis of plot for spectral plotting
  options: (list) [min,max]
  default: [] (full range)
  example: flrange=[-0.1,2.0] if 'K'
           assumes current fluxunit
sprange -- range for spectral axis of plot
  options: (list) [min,max]
  default: [] (full range)
  example: sprange=[42.1,42.5] if 'GHz'
           assumes current specunit
linecat -- control for line catalog plotting for spectral plotting
  options: (str) 'all','none' or by molecule
  default: 'none' (no lines plotted)
  example: linecat='SiO' for SiO lines
           linecat='*OH' for alcohols

```

```

        uses sprange to limit catalog
WARNING: specunit must be in frequency (*Hz)
        to plot from the line catalog!
        and must be 'GHz' or 'MHz' to use
        sprange to limit catalog
linedop -- doppler offset for line catalog plotting (spectral plotting)
options: (float) doppler velocity (km/s)
default: 0.0
example: linedop=-30.0
colormap -- the colours to be used for plot lines.
default: None
example: colormap="green red black cyan magenta" (html standard)
        colormap="g r k c m" (abbreviation)
        colormap="#008000 #00FFFF #FF0090" (RGB tuple)
        The plotter will cycle through these colours
        when lines are overlaid (stacking mode).
linestyles -- the linestyles to be used for plot lines.
default: None
example: linestyles="line dashed dotted dashdot dashdotdot dashdashdot".
        The plotter will cycle through these linestyles
        when lines are overlaid (stacking mode).
        WARNING: Linestyles can be specified only one color has been set.
linewidth -- width of plotted lines.
default: None
example: linewidth=1 (integer)
        linewidth=0.75 (double)
histogram -- plot histogram
options: (bool) True, False
default: False
plotfile -- file name for hardcopy output
options: (str) filename.eps,.ps,.png
default: '' (no hardcopy)
example: 'specplot.eps','specplot.png'
        Note this autodetects the format from
        the suffix (.eps,.ps,.png).
overwrite -- overwrite the output file if already exists
options: (bool) True,False
default: False

```

DESCRIPTION:

Task `sdplot` displays single-dish spectra. It assumes that the spectra have been calibrated in `sdcal`. It does allow selection of scans, IFs, polarizations, and some time and channel averaging/smoothing options also, but does not write data.

*** Only apply to 'spectra' plotttype ***

Some header information of the data are plotted at top. Note that `colormap` and `linestyles` cannot be controlled at this time. `'linestyles'` is ignored if both of them are specified. Some plot options, like annotation and changing titles, legends, fonts, and the like are not supported in this task. You should use `sd.plotter` from the ASAP toolkit directly for this.

This task uses the JPL line catalog as supplied by ASAP. If you wish to use a different catalog, or have it plot the line IDs from top or bottom (rather than alternating), then you will need to explore the `sd` toolkit also. See § A.3.9 for more information.

Note that multiple scans and IFs can in principle be handled through stacking and paneling, but this is fairly rudimentary at present and you have little control of what happens in individual panels. We recommend that you use `scanlist`, `field`, and `iflist` to give a single selection for each run.

After plotting, you can display a spectral value on a pop-up window along with mouse movement. Click the LEFT-mouse button over a spectrum to select it, and drag mouse in the panel to output its value at the x-position of mouse cursor. The selection is released when you release the mouse button.

The task `sdplot` adds an additional toolbar to ASAP plotter which has three buttons, `'spec value'`, `'statistics'`, and `'Quit'`. When the `'spec value'` button is pressed and activated, you can display a spectral value on the toolbar along with mouse movement. Click the LEFT-mouse button over a spectrum to select it, and drag mouse in the panel to output its value at the x-position of mouse cursor. The selection is released when you release the mouse button. When `'statistic'` button is pressed and activated, you can get statistic values of a channel region selected by mouse. Select a rectangle region with LEFT/RIGHT-mouse buttons to SELECT/EXCLUDE the channel region to calculate statistics for the all spectra plotted. The results are printed on the casapy console. Press `'Quit'` button to close the plotter.

*** other `plottype` options ***

`plottype='totalpower'` is used to plot the total power data. and only plot option is amplitude versus data row number. `plottype='azel'` plots azimuth and elevation tracks of the source. `plottype='pointing'` plots antenna pointing. Currently most of the plotting parameters are ignored for these modes.

See the `sdaverage` description for information on `fluxunit` conversion and the `telescopeparm` parameter.

WARNING: be careful plotting OTF (on-the-fly) mosaic data with lots of fields!

A.2.1.12 `sdsave`

Keyword arguments:

```
sdfile -- name of input SD dataset
antenna -- antenna name or id (only effective for MS input).
rowlist -- list of row numbers to process
          default: [] (use all rows)
```

```

example: [0,2,4,6]
    For expert users only!
    this selection is applied first, and then
    followed by the selection with scans, fields,
    ifs, and polarizations.
scanlist -- list of scans to process
    default: [] (use all scans)
    example: [21,22,23,24]
        this selection is in addition to field,
        iflist, and pollist
field -- selection string for selecting scans by name
    default: '' (no name selection)
    example: 'FLS3a*'
        this selection is in addition to scanlist,
        iflist, and pollist
iflist -- list of IF id numbers to select
    default: [] (use all IFs)
    example: [15]
        this selection is in addition to scanlist,
        field, and pollist
pollist -- list of polarization id numbers to select
    default: [] (use all polarizations)
    example: [1]
        this selection is in addition to scanlist,
        field, and iflist
scanaverage -- average integrations within scans
    options: (bool) True,False
    default: False
    example: if True, average integrations before it is saved
timeaverage -- average times for multiple scan cycles
    options: (bool) True,False
    default: False

>>>timeaverage expandable parameter
    tweight -- weighting for time average
        options: 'none'
                'var' (1/var(spec) weighted)
                'tsys' (1/Tsys**2 weighted)
                'tint' (integration time weighted)
                'tintsys' (Tint/Tsys**2)
                'median' (median averaging)
        default: 'none'

polaverage -- average polarizations
    options: (bool) True,False

```

```

    default: False

>>>polaverage expandable parameter
    pweight -- weighting for polarization average
            options: 'none'
                   'var' (1/var(spec) weighted)
                   'tsys' (1/Tsys**2 weighted)

outfile -- name of output dataset
         default: ''
outform -- output data format
         default: 'ASAP'
         Options: 'ASAP', 'MS2', 'SDFITS', 'ASCII'
overwrite -- overwrite the output file if already exists
           options: (bool) True,False
           default: False
           WARNING: if outform='ASCII', this parameter is ignored

```

DESCRIPTION:

Task `sdsave` writes the single dish data to a disk file in specified format (ASAP, MS2, SDFITS, ASCII). It is possible to save the subset of the data by selecting row numbers, scan numbers, IF ids and field names. The ASAP (scantable) format is recommended for further analysis using `sd` tool. For further imaging using `imager`, save the data to the Measurement Set (MS2).

WARNING: The parameter `rowlist` enables you to make data selection based on row number in the Measurement Set or scantable for data saving. Note that data should be treated carefully when applying row based selection, since row numbers can be changed easily by sorting, prior selection, etc. Therefore, this parameter is expected to be used by expert users only .

A.2.1.13 sdscale

Keyword arguments:

```

sdfile -- name of input SD dataset
antenna -- antenna name or id (only effective for MS input).
factor -- scaling factor, float or one- or two-dimensional float list
         default: 1 (no scaling)
scaletsys -- scaling of associated Tsys
           default: False
outfile -- output file name
         outfile='' will write the data to a file named,
         <sdfile>_scaled<factor>
         default: ''
overwrite -- overwrite the output file if already exists
           options: (bool) True,False

```


default: False

DESCRIPTION:

Task `sdscale` performs scaling of single-dish spectra. By setting `scaletsys = True`, associated `Tsys` is also scaled. `Tsys` information are written into the file `'sdscale.log'` as well as they are displayed in the terminal window. The `infile` can be any of `ASAP`, `MS`, `SDFITS`, or `RPFITS` format. If `outfile` name is given or `outfile=''` (default), the scaled data is written to a new file with the same format as the input data (Note: in case of the `RPFITS` format input data, it will be written to `SDFITS` format).

The scaling factor, `factor`, accepts both scalar type and list type value. The list must be one or two dimensional. If `factor` is one dimensional, its length must coincide with a number of spectral channel. If `factor` is two dimensional, its shape must be $(n,1)$ or (n,m) , where n is a number of spectrum, while m is a number of channel for each spectrum. m can be variable for each spectrum. In addition, the `factor` can be an ASCII filename that stores a space-separated list of scaling factor consisting of adequate number of rows and columns. For example, if the content of input ASCII file is shown as,

```
0.5 0.3 0.2 1.0 0.2 0.9
```

it is interpreted as a list `[[0.5,0.3,0.2],[1.0,0.2,0.9]]`.

A.2.1.14 `sdstat`

Keyword arguments:

`sdfile` -- name of input SD dataset

default: none - must input file name

example: `'mysd.asap'`

See `sdcal` for allowed formats.

`antenna` -- antenna name or id (only effective for `MS` input).

`fluxunit` -- units for line flux

options: (str) `'K','Jy',''`

default: `''` (keep current fluxunit)

WARNING: For `GBT` data, see description below.

>>> `fluxunit` expandable parameter

`telescopeparm` -- the telescope characteristics

options: (str) name or (list) list of gain info

default: `''` (none set)

example: if `telescopeparm=''`, it tries to get the telescope name from the data.

Full antenna parameters (diameter,ap.eff.) known to `ASAP` are

`'ATPKSMB', 'ATPKSHOH', 'ATMOPRA', 'DSS-43',`

`'CEDUNA','HOBART'`. For `GBT`, it fixes default fluxunit

to `'K'` first then convert to a new fluxunit.

```

telescopeparm=[104.9,0.43] diameter(m), ap.eff.
telescopeparm=[0.743] gain in Jy/K
telescopeparm='FIX' to change default fluxunit
see description below

```

```

specunit -- units for spectral axis
  options: (str) 'channel','km/s','GHz','MHz','kHz','Hz',''
  default: '' (=current)
frame -- frequency frame for spectral axis
  options: (str) 'LSRK','REST','TOPO','LSRD','BARY',
              'GEO','GALACTO','LGROUP','CMB'
  default: currently sets frame in scantable
  WARNING: frame='REST' not yet implemented
doppler -- doppler mode
  options: (str) 'RADIO','OPTICAL','Z','BETA','GAMMA'
  default: currently sets doppler in scantable
scanlist -- list of scan numbers to process
  default: [] (use all scans)
  example: [21,22,23,24]
field -- selection string for selecting scans by name
  default: '' (no name selection)
  example: 'FLS3a*'
           this selection is in addition to scanlist
           iflist, and pollist
iflist -- list of IF id numbers to select
  default: [] (use all IFs)
  example: [15]
           this selection is in addition to field, scanlist
           and pollist
pollist -- list of polarization id numbers to select
  default: [] (use all pols)
  example: [1]
           this selection is in addition to field, scanlist
           and iflist
masklist -- list of mask regions to INCLUDE in stats
  default: [] (whole spectrum)
  example: [4000,4500] for one region
           [[1000,3000],[5000,7000]]
           these must be pairs of [lo,hi] boundaries
invertmask -- invert mask (EXCLUDE masklist instead)
  options: (bool) True,False
  default: false
interactive -- determines interactive masking
  options: (bool) True,False
  default: False

```

```

    example: interactive=True allows adding and deleting mask
            regions by drawing rectangles on the plot with mouse.
            Draw a rectangle with LEFT-mouse to ADD the region to
            the mask and with RIGHT-mouse to DELETE the region.
statfile -- name of output file for line statistics
    default: '' (no output statistics file)
    example: 'stat.txt'
format -- format string to print statistic values
    default: '3.3f'
overwrite -- overwrites the statistics file if already exists
    options: (bool) True,False
    default: False

```

```

Returns: a Python dictionary of line statistics
    keys: 'rms', 'stddev', 'max', 'max_abscissa', 'min',
          'min_abscissa', 'sum', 'median', 'mean',
          'totint', 'eqw',
    example: xstat=sdstat(); print "rms = ",xstat['rms']
            these can be used for testing in scripts or
            for regression

```

```

    'max_abscissa' and 'min_abscissa' refer to the abscissa
    (channel/frequency/velocity) of max and min intensity.
    'totint' is the integrated intensity (sum*dx)
    where dx is the abscissa interval in 'specunit'.
    'eqw' is equivalent width (totint/mag) where mag
    is either max or min depending on which has
    greater magnitude.
    Note that both 'totint' and 'eqw' are quantities
    (python dictionaries with keys, 'unit' and 'value').

```

DESCRIPTION:

Task `sdstat` computes basic statistics (rms,mean,median,sum) for single-dish spectra. It assumes that the spectra have been calibrated. Furthermore, it assumes that any time and channel averaging/smoothing has also already been done as there are no controls for these. Note that you can run `sdcal` with `calmode='none'` and do selection, writing out a new scantable. The calculated statistics are written into a file specified by `statfile`. Interactive mask specification is possible with `interactive=True`. Integrated intensity will be shown on the screen and will be included in the saved statfile (but not yet available in the returned dictionary).

Note that multiple scans and IFs can in principle be handled, but we recommend that you use `scanlist`, `field`, `iflist`, and `pollist` to give a single selection for each run.

See the `sdcal` description for information on the `fluxunit` conversion and the `telescopeparm` parameter.

WARNING: If you do have multiple scantable rows, then `xstat` values will be lists.

A.2.1.15 `sdtipimaging`

Keyword arguments:

`sdfile` -- name of input SD (MS) dataset

`calmode` -- calibration mode (currently only baseline subtraction)

options: 'baseline', 'none'

default: 'none'

example: choose mode 'none' if you have
already calibrated and want to do
plotting nd/or imaging

>>> `calmode='baseline'` expandable parameters

`masklist` -- mask in numbers of rows from each edge of each scan to
be included for baseline fitting

default: none

example: [30,30] or [30]

used first 30 rows and last 30 rows of each scan for
the baseline

`blpoly` -- polynomial order for the baseline fit

default: 1

`flaglist` -- list of scan numbers to flag (ranges can be accepted)

default: [] (use all scans)

example: [[0,3],80]

flag the scan range [0,3] = [0,1,2,3] and scan 80

`antenna` -- select data based on antenna name(s) or id(s) in string

default: '' (use all antennas)

example: '0,1', 'DV01'

WARNING: currently baseline subtraction properly
only one of the antennas.

`stokes` -- select data based on stokes or polarization type

default: '' (use all polarizations)

example: 'XX'

`createimage` -- do imaging?

default: False

>>> `createimage=True` expandable parameters

`imagenname` -- output image name

default: none

example: 'mySDimage.im'

```

imsize -- x and y image size in pixels, symmetric for single value
        default: [256,256]
        example: imsize=200 (equivalent to [200,200])
cell -- x and y cell size. default unit arcmin
        default: '1.0arcmin'
        example: cell=['0.2arcmin, 0.2arcmin']
               cell='0.2arcmin' (equivalent to example above)
phasecenter -- image phase center: direction measure or fieldid
        default: 0
        example: 'J2000 13h44m00 -17d02m00', 'AZEL -123d48m29 15d41m41'
ephemsrcname -- ephemeris source name to proper shifting to center on
               the moving source for imaging
        default: ''
               if the source name in the data matches one of the known
               solar objects by the system, this task automatically set
               the source name.
        example: 'moon'
pointingcolumn -- pointing data column to use
        option: 'direction', 'target', 'pointing_offset', 'source_offset', 'encoder'
        default: 'direction'
gridfunction -- gridding function for imaging
        options: 'BOX' (Box-car), 'SF' (Spheroidal), 'PB' (Primary-beam)
        default: 'BOX'
        example: 'SF'

plotlevel -- control for plotting of results
        options: (int) 0=none, 1=some, 2=more, <0=hardcopy
        default: 0 (no plotting)
        example: plotlevel<0 as abs(plotlevel), e.g.
               -1: hardcopy plot (will be named <sdfile>_scans.eps)
                  1: plot raw data, calibrated data (for calmode='baseline')
                     plot raw or if exist calibrated data (for calmode='none')
                  2: plot raw data, progressively display baseline fitting for each
                     scan, and final calibrated data (for calmode='baseline')

```

DESCRIPTION:

Task `sdtipimaging` performs data selection, calibration, and imaging for single-dish totalpower raster scan data. This is a still experimental task made to work for the data taken at the ALMA Testing Facility (ATF) and OSF. Currently, this task directly accesses the Measurement Set data because of the data access efficiency. So it differs from other single-dish tasks that mostly operate on the ASAP scantable data format. By setting `calmode='none'`, one can run `sdtipimaging` to plot the data (raw or calibrated, if exists) and further imaging by setting `createimage=True`. The calibration available at this moment is just a simple baseline subtraction for each scan. The fitted

regions set by `masklist` are the common for all the scans. Selection of the antennas can be made by setting antenna ID(s) or antenna name(s) in string (e.g. '0', '0,1', 'DV01',etc.). For baseline subtraction, it currently works properly for a single antenna selection. So a separate `sdtipimaging` task needs to be run for each antenna. It currently assumes that the data has a single `spw(=0)` and `fieldid(=0)`. By setting `flaglist`, one can set flag by scan numbers to be excluded from imaging. (Note: 'scan numbers' are determined from state id and related to SUB_SCAN column in STATE subtable and they are typically different from SCAN_NUMBER in MS.) The selection of polarizations can be made by specifying the polarization name in stokes, such as 'XX' or 'YY' for linear polarizations. For example, with `createimage=True`, `stokes='XXYY'` will produces an image cube with each plane contains the image of one of the polarizations while `stokes=''` or `stokes='I'` will produces a 'total intensity' or Stokes I image. Among the imaging sub-parameters, `ephemsrcname` is used to set the name of a moving source such as planets to produce a stationary image (can be omitted), and `pointingcolumn` is used to specify which pointing data column to use for imaging.

A.2.1.16 `sdimprocess`

Keyword arguments:

`sdimages` -- name of input SD (FITS or CASA) image

`mode` -- processing mode

default: 'basket'

options: 'basket', 'press'

>>>mode expandable parameters

`direction` -- scan direction in unit of degree

default: []

example: [0.0,90.0]

`masklist` -- mask width for Basket-Weaving on percentage

default: 1.0 (1.0% of map size)

`numpoly` -- order of polynomial fit in Presssed-out

default: 2

`beamsize` -- beam size

default: 0.0

example: 10.0 (interpreted as '10arcsec'), '1arcmin'

`smoothsize` -- smoothing beam in Presssed-out

default: 2.0 (interpreted as 2.0 * beamsize)

example: '1arcmin' (set smoothsize directly)

`tmax` -- maximum value used for process

default: 0.0 (no threshold in maximum)

example: 10.0 (mask data larger value than 10.0)

`tmin` -- minimum value used for process

default: 0.0 (no threshold in minimum)

example: -10.0 (mask data smaller value than -10.0)

```

imagename -- output CASA image name
            default: '' (use default name)
            example: 'output.im'
overwrite -- overwrite option for imagename
            default: False (not overwrite)
            options: True, False
            example: if True, existing file will be overwritten

```

DESCRIPTION:

Task `sdimprocess` is used to remove a scanning noise that appears as a striped noise pattern along the scan direction in a raster scan data.

By default, the scanning noise is removed by using the 'Basket-Weaving' method (Emerson & Grave 1988) that requires multiple images that observed exactly the same area with different scanning direction. If only one image is available, the 'Pressed-out' method (Sofue & Reich 1979) can be used to remove the scanning effect.

For 'Basket-Weaving', scanning directions must have at least two different values. Normally, the scanning direction should be specified for each input image. Otherwise, specified scanning directions will be used iteratively. The `masklist` is a width of masking region in the Fourier plane. It is specified as a fraction (percentage) of the image size.

For 'Pressed-out', the scanning direction must be unique. There are two ways to specify a size of smoothing beam used for process. One is to specify smoothing size directly, where `smoothsize` is specified as string that consists of a numerical value and an unit (e.g. '10.0arcsec'). The value of `beamsize` will be ignored in this case. Another way to specify smoothing size is to set an observed beam size and indicate them smoothing size as a scale factor of the observed beam. In this case, the `beamsize` is interpreted as the observed beam size, and the `smoothsize` is the scale factor. If the `beamsize` is provided as float value, its unit is assumed to have 'arcsec' units. It is also possible to set the `beamsize` as string consisting of the numerical value and the unit. The `smoothsize` must be float value.

The `sdimages` only allows an image data (CASA or FITS), and does not work with MS or Scantable. The `direction` is an angle with respect to the horizontal direction in degree units. Any value may be interpreted properly, but the value ranging from 0.0 to 180.0 will be secure. The `tmax` and the `tmin` is used to specify a threshold that defines a range of spectral values used for processing. The data point that has the value larger than `tmax` or smaller than `tmin` will be excluded from the processing. The default (0.0) is no threshold. The `imagename` specifies an output CASA image name. If the `imagename` is empty, the default name ('sdimprocess.out.im') will be used.

A.2.1.17 `sdsim`

Task `sdsim` simulates single dish observations. Note this is a preliminary prototype task and functionalities are limited. New functionality is actively being added, so if you have changed versions of CASA, check the inputs carefully.

single dish simulation task (prototype):

This task simulates single dish observations. Note this is a preliminary prototype task and functionalities are limited. New functionality is actively being added, so if you have changed versions of CASA, check the inputs carefully.

Keyword arguments:

-- Input Model:

modelimage -- Name of an image to simulate.

inbright -- Peak surface brightness to scale input image
in Jy/square arcsec.
options: 'default' or surface brightness in Jy/sq.
default: 'default'

ignorecoord -- ignore whatever is in the image header for coordinates
options: (bool) True,False
default: False

antennalist -- Name of ascii file containing antenna positions.
each row has x y z coordinates and antenna diameter;
header lines are required to specify the observatory name
and coordinate system e.g.
observatory=ALMA
coordsys=UTM
datum=WGS84
zone=19

antenna -- Antenna id with which observed.
default: ''

-- Output Control:

project -- Name of project simulated; created ms and images will
start with this string.
default: 'simsd'.

startfreq -- Frequency of first channel
default: '89GHz'

chanwidth -- Channel width
default: '10MHz'

refdate -- Central time of simulated observation.
default: '2012/05/21/22:05:00'

[alpha] ** observations are centered at the nearest transit **

totaltime --- total time of observation e.g '7200s'

integration --- Time interval for each integration
default: '10s'

scanlength --- number of integrations per pointing
default: 5


```

direction -- image center direction
    * can optionally be a list of pointings or
    a filename which stores a list of pointings, which will
    override pointingspacing.
    Otherwise sdsim will automatically calculates pointings
    in regular lattice grid. When direction is a
    list, the centroid of direction will be used as the center.
    default: ['J2000 19h00m00 -40d00m00']
pointingspacing-- spacing in between beams
    default: '1arcmin'
mosaicsize --- angular size of desired area to map
    default: ['1.0arcmin', '1.0arcmin']
cell -- output cell/pixel size
    default: '0.1arcsec'
imsize -- output image size in pixels (x,y)
    options: (list)
    default: [128,128]
stokes -- Stokes parameters to image
    default: 'I'

```

```

-----
-- Corrupting data:

```

```

noise_thermal -- add thermal noise
    options: (bool) True, False
    default: False
    * [alpha] currently only knows about ALMA receivers
      noise_thermal expandable parameters
t_sky -- atmospheric temperature in K
    default: 260.0
tau0 -- zenith opacity at observing frequency
    default: 0.1

```

```

-----
How to specify a model image:
-----

```

* sdsim requires a CASA or fits image. If you merely have a grid of numbers, you will need to write them out as fits or write a CASA script to read them in and use the ia tool to create an image and insert the data.

* sdsim does NOT require a coordinate system in the header. If the coordinate information is incomplete, missing, or you would like to override it, set "ignorecoord=True". sdsim will then assume that the axes of your input correspond to RA, Dec, and (optionally) frequency and (optionally) Stokes parameter. It will assume that the spatial pixel

size equals the output parameter "cell", the center of the image is in the (average of the) output parameter "direction", the spectral reference frequency and starting frequency equal the output parameter "startfreq", and the channel width equals the output parameter "chanwidth".

```
-----
Output produced:
-----
```

```
$project.$modelimage.coord = input image recast into 4d image
$project.$modelimage.flat  = input image regridded to size, moment zero
                             image if the input was a cube.
$project.ms = synthesized visibilities
$project.noisy.ms = visibilities with noise added
    ** if noise is requested, then the noisy visibilities will be
        used to create the synthesized image.
$project.im = output image created from $project.noisy.ms (if exists) or
              $project.ms.
```

R3.0.2 New Features: Single dish simulation can now be done in the same task as interferometric, and the simulated datasets can be jointly imaged, and the combined image analyzed. See the `simdata2` chapter, Sec. B.

A.2.2 Single Dish Analysis Use Cases With SDTasks

A.2.2.1 GBT Position Switched Data Analysis

As an example, the following illustrates the use of the SDtasks for the Orion data set, which contains the HCCCN line in one of its IFs. This walk-through contains comments about setting parameter values and some options during processing.

```
#####
#
# ORION-S SDtasks Use Case
# Position-Switched data
# Version TT 2008-10-14 (updated)
# Version STM 2007-03-04
#
# This is a detailed walk-through
# for using the SDtasks on a
# test dataset.
#
#####
import time
```

```
import os

# NOTE: you should have already run
# asap_init()
# to import the ASAP tools as sd.<tool>
# and the SDtasks

#
# This is the environment variable
# pointing to the head of the CASA
# tree that you are running
casapath=os.environ['AIPSPATH']

#
# This bit removes old versions of the output files
os.system('rm -rf sdusecase_orions* ')
#
# This is the path to the OrionS GBT ms in the data repository
datapath=casapath+'/data/regression/ATST5/OrionS/OrionS_rawACSmod'
#
# The following will remove old versions of the data and
# copy the data from the repository to your
# current directory. Comment this out if you already have it
# and don't want to recopy
os.system('rm -rf OrionS_rawACSmod')
copystring='cp -r '+datapath+' .'
os.system(copystring)

# Now is the time to set some of the more useful
# ASAP environment parameters (the ones that the
# ASAP User Manual claims are in the .asaprc file).
# These are in the Python dictionary sd.rcParams
# You can see whats in it by typing:
#sd.rcParams
# One of them is the 'verbose' parameter which tells
# ASAP whether to spew lots of verbiage during processing
# or to keep quiet. The default is
#sd.rcParams['verbose']=True
# You can make ASAP run quietly (with only task output) with
#sd.rcParams['verbose']=False

# Another key one is to tell ASAP to save memory by
# going off the disk instead. The default is
#sd.rcParams['scantable.storage']='memory'
# but if you are on a machine with small memory, do
```

```

#sd.rcParams['scantable.storage']='disk'

# You can reset back to defaults with
#sd.rcdefaults

#####
#
# ORION-S HC3N
# Position-Switched data
#
#####
startTime=time.time()
startProc=time.clock()

#####
# List data
#####
# List the contents of the dataset
# First reset parameter defaults (safe)
default('sdlist')

# You can see its inputs with
#inp('sdlist')
# or just
#inp
# now that the defaults('sdlist') set the
# taskname='sdlist'
#
# Set the name of the GBT ms file
sdfile = 'OrionS_rawACSmod'

# Set an output file in case we want to
# refer back to it
listfile = 'sdusecase_orions_summary.txt'
sdlist()

# You could also just type
#go

# You should see something like:
#
#-----
# Scan Table Summary
#-----
#Beams:          1

```

```

#IFs:          26
#Polarisations: 2 (linear)
#Channels:     8192
#
#Observer:     Joseph McMullin
#Obs Date:     2006/01/19/01:45:58
#Project:      AGBT06A_018_01
#Obs. Type:    OffOn:PSWITCHOFF:TPWCAL
#Antenna Name: GBT
#Flux Unit:    Jy
#Rest Freqs:   [4.5490258e+10] [Hz]
#Abcissa:      Channel
#Selection:    none
#
#Scan Source   Time      Integration
#   Beam      Position (J2000)
#           IF      Frame  RefVal      RefPix      Increment
#-----
# 20 OrionS_psr 01:45:58  4 x      30.0s
#   0 05:15:13.5 -05.24.08.2
#   0 LSRK 4.5489354e+10 4096 6104.233
#   1 LSRK 4.5300785e+10 4096 6104.233
#   2 LSRK 4.4074929e+10 4096 6104.233
#   3 LSRK 4.4166215e+10 4096 6104.233
# 21 OrionS_ps 01:48:38  4 x      30.0s
#   0 05:35:13.5 -05.24.08.2
#   0 LSRK 4.5489354e+10 4096 6104.233
#   1 LSRK 4.5300785e+10 4096 6104.233
#   2 LSRK 4.4074929e+10 4096 6104.233
#   3 LSRK 4.4166215e+10 4096 6104.233
# 22 OrionS_psr 01:51:21  4 x      30.0s
#   0 05:15:13.5 -05.24.08.2
#   0 LSRK 4.5489354e+10 4096 6104.233
#   1 LSRK 4.5300785e+10 4096 6104.233
#   2 LSRK 4.4074929e+10 4096 6104.233
#   3 LSRK 4.4166215e+10 4096 6104.233
# 23 OrionS_ps 01:54:01  4 x      30.0s
#   0 05:35:13.5 -05.24.08.2
#   0 LSRK 4.5489354e+10 4096 6104.233
#   1 LSRK 4.5300785e+10 4096 6104.233
#   2 LSRK 4.4074929e+10 4096 6104.233
#   3 LSRK 4.4166215e+10 4096 6104.233
# 24 OrionS_psr 02:01:47  4 x      30.0s
#   0 05:15:13.5 -05.24.08.2
#   12 LSRK 4.3962126e+10 4096 6104.2336

```

```

#           13      LSRK    4.264542e+10   4096   6104.2336
#           14      LSRK    4.159498e+10   4096   6104.2336
#           15      LSRK    4.3422823e+10  4096   6104.2336
# 25 OrionS_ps      02:04:27    4 x      30.0s
#           0      05:35:13.5 -05.24.08.2
#           12      LSRK    4.3962126e+10  4096   6104.2336
#           13      LSRK    4.264542e+10  4096   6104.2336
#           14      LSRK    4.159498e+10  4096   6104.2336
#           15      LSRK    4.3422823e+10  4096   6104.2336
# 26 OrionS_psr     02:07:10    4 x      30.0s
#           0      05:15:13.5 -05.24.08.2
#           12      LSRK    4.3962126e+10  4096   6104.2336
#           13      LSRK    4.264542e+10  4096   6104.2336
#           14      LSRK    4.159498e+10  4096   6104.2336
#           15      LSRK    4.3422823e+10  4096   6104.2336
# 27 OrionS_ps      02:09:51    4 x      30.0s
#           0      05:35:13.5 -05.24.08.2
#           12      LSRK    4.3962126e+10  4096   6104.2336
#           13      LSRK    4.264542e+10  4096   6104.2336
#           14      LSRK    4.159498e+10  4096   6104.2336
#           15      LSRK    4.3422823e+10  4096   6104.2336

```

```

# The HC3N and CH3OH lines are in IFs 0 and 2 respectively
# of scans 20,21,22,23. We will pull these out in our
# calibration.

```

```
#####
```

```
# Calibrate data
```

```
#####
```

```
# We will use the sdcal task to calibrate the data.
```

```
# Set the defaults
```

```
default('sdcal')
```

```
# You can see the inputs with
```

```
#inp
```

```
# Set our sfile (which would have been set from our run of
```

```
# sdlist if we were not cautious and reset defaults).
```

```
sfile = 'OrionS_rawACSmod'
```

```
fluxunit = 'K'
```

```
# Lets leave the spectral axis in channels for now
```

```
specunit = 'channel'
```

```
# This is position-switched data so we tell sdcal this
```

```
calmode = 'ps'

# For GBT data, it is safest to not have scantable pre-average
# integrations within scans.
average = True
scanaverage = False

# We do want sdcals to average up scans and polarization after
# calibration however. The averaging of scans are weighted by
# integration time and Tsys, and the averaging of polarization
# by Tsys.
timeaverage = True
tweight = 'tintsys'
polaverage = True
pweight = 'tsys'
# Do an atmospheric optical depth (attenuation) correction
# Input the zenith optical depth at 43 GHz
tau = 0.09

# Select our scans and IFs (for HC3N)
scanlist = [20,21,22,23]
iflist = [0]

# We do not require selection by field name (they are all
# the same except for on and off)
field = ''

# We will do some spectral smoothing
# For this demo we will use boxcar smoothing rather than
# the default
#kernel='hanning'
# We will set the width of the kernel to 5 channels
kernel = 'boxcar'
kwidth = 5

# We wish to fit out a baseline from the spectrum
# The GBT has particularly nasty baselines :(
# We will let ASAP use auto_poly_baseline mode
# but tell it to drop the 1000 edge channels from
# the beginning and end of the spectrum.
# A 2nd-order polynomial will suffice for this test.
# You might try higher orders for fun.
blmode = 'auto'
blpoly = 2
edge = [1000]
```

```
# We will not give it regions as an input mask
# though you could, with something like
#masklist=[[1000,3000],[5000,7000]]
masklist = []

# By default, we will not get plots in sdcal (but
# can make them using sdplot).
plotlevel = 0
# But if you wish to see a final spectrum, set
#plotlevel = 1
# or even
#plotlevel = 2
# to see intermediate plots and baselining output.

# Now we give the name for the output file
outfile = 'sdusecase_orions_hc3n.asap'

# We will write it out in ASAP scantable format
outform = 'asap'

# You can look at the inputs with
#inp

# Before running, lets save the inputs in case we want
# to come back and re-run the calibration.
saveinputs('sdcal','sdcal.orions.save')
# These can be recovered by
#execfile 'sdcal.orions.save'

# We are ready to calibrate
sdcal()

# Note that after the task ran, it produced a file
# sdcal.last which contains the inputs from the last
# run of the task (all tasks do this). You can recover
# this (anytime before sdcal is run again) with
#execfile 'sdcal.last'

#####
# List data
#####
# List the contents of the calibrated dataset
# Set the input to the just created file
sdfile = outfile
```



```

listfile = ''
sdlist()

# You should see:
#
#-----
# Scan Table Summary
#-----
#Beams:          1
#IFs:           26
#Polarisations: 1 (linear)
#Channels:      8192
#
#Observer:      Joseph McMullin
#Obs Date:      2006/01/19/01:45:58
#Project:       AGBT06A_018_01
#Obs. Type:     OffOn:PSWITCHOFF:TPWCAL
#Antenna Name:  GBT
#Flux Unit:     K
#Rest Freqs:    [4.5490258e+10] [Hz]
#Abcissa:       Channel
#Selection:     none
#
#Scan Source      Time      Integration
#  Beam  Position (J2000)
#      IF      Frame  RefVal      RefPix  Increment
#-----
#  0 OrionS_ps    01:52:05    1 x    08:00.5
#      0    05:35:13.5 -05.24.08.2
#      0      LSRK    4.5489354e+10    4096    6104.233
#
# Note that our scans are now collapsed (timeaverage=True) but
# we still have our IF 0

#####
# Plot data
#####
default('sdplot')

# The file we produced after calibration
# (if we hadn't reset defaults it would have
# been set - note that sdplot,sdfit,sdstat use
# sdfile as the input file, which is the output
# file of sdcal).
sdfile = 'sdusecase_orions_hc3n.asap'

```

```
# Lets just go ahead and plot it up as-is
sdplot()

# Looks ok. Plot with x-axis in GHz
specunit='GHz'
sdplot()

# Note that the rest frequency in the scantable
# is set correctly to the HCCCN line at 45.490 GHz.
# So you can plot the spectrum in km/s
specunit='km/s'
sdplot()

# Zoom in
sprange=[-100,50]
sdplot()

# Lets plot up the lines to be sure
# We have to go back to GHz for this
# (known deficiency in ASAP)
specunit='GHz'
sprange=[45.48,45.51]
linecat='all'
sdplot()

# Too many lines! Focus on the HC3N ones
linecat='HCCCN'
sdplot()

# Finally, we can convert from K to Jy
# using the aperture efficiencies we have
# coded into the sdtasks
# For GBT data, do not set telescopeparm
fluxunit='Jy'
telescopeparm=''
sdplot()

# Lets save this plot
plotfile='sdusecase_orions_hc3n.eps'
sdplot()

#####
# Off-line Statistics
#####
```

```
# Now do some region statistics
# First the line-free region
# Set parameters
default('sdstat')
sdfile = 'sdusecase_orions_hc3n.asap'

# Keep the default spectrum and flux units
# K and channel
fluxunit = ''
specunit = ''

# Pick out a line-free region
# You can bring up a default sdplot again
# to check this
masklist = [[5000,7000]]

# This is a line-free region so we don't need
# to invert the mask
invertmask = False

# You can check with
#inp

# sdstat returns some results in
# the Python dictionary. You can assign
# this to a variable
off_stat=sdstat()

# and look at it
off_stat
# which should give
# {'eqw': 38.563105620704945,
#  'max': 0.15543246269226074,
#  'mean': -0.0030361821409314871,
#  'median': -0.0032975673675537109,
#  'min': -0.15754437446594238,
#  'rms': 0.047580458223819733,
#  'stddev': 0.047495327889919281,
#  'sum': -6.0754003524780273}

#You see it has some keywords for the various
#stats. We want the standard deviation about
#the mean, or 'stddev'
print "The off-line std. deviation = ",off_stat['stddev']
```

```

# which should give
# The off-line std. deviation = 0.0474953278899

# or better formatted (using Python I/O formatting)
print "The off-line std. deviation = %5.3f K" %\
      (off_stat['stddev'])
# which should give
# The off-line std. deviation = 0.047 K

#####
# On-line Statistics
#####
# Now do the line region
# Continue setting or resetting parameters
masklist = [[3900,4200]]

line_stat = sdstat()

# look at these
line_stat
# which gives
# {'eqw': 73.335154614280981,
#  'max': 0.92909121513366699,
#  'mean': 0.22636228799819946,
#  'median': 0.10317134857177734,
#  'min': -0.13283586502075195,
#  'rms': 0.35585442185401917,
#  'stddev': 0.27503398060798645,
#  'sum': 68.135047912597656}

# of particular interest are the max value
print "The on-line maximum = %5.3f K" % (line_stat['max'])
# which gives
# The on-line maximum = 0.929 K

# and the estimated equivalent width (in channels)
# which is the sum/max
print "The estimated equivalent width = %5.1f channels" %\
      (line_stat['eqw'])
# which gives
# The estimated equivalent width = 73.3 channels

#####
# Line Fitting
#####

```

```
# Now we are ready to do some line fitting
# Default the parameters
default('sdfit')

# Set our input file
sdfile = 'sdusecase_orions_hc3n.asap'

# Stick to defaults
# fluxunit = 'K', specunit = 'channel'
fluxunit = ''
specunit = ''

# We will try auto-fitting first
fitmode = 'auto'
# A single Gaussian
nfit = [1]
# Leave the auto-parameters to their defaults for
# now, except ignore the edge channels
edge = [1000]

# Lets see a plot while doing this
plotlevel = 1

# Save the fit output in a file
fitfile = 'sdusecase_orions_hc3n.fit'

# Go ahead and do the fit
fit_stat=sdfit()

# If you had verbose mode on, you probably saw something
# like:
#
# 0: peak = 0.811 K , centre = 4091.041 channel, FWHM = 72.900 channel
#    area = 62.918 K channel
#

# The fit is output in the dictionary

fit_stat
#
# {'cent': [[4091.04052734375, 0.72398632764816284]],
#  'fwhm': [[72.899894714355469, 1.7048574686050415]],
#  'nfit': 1,
#  'peak': [[0.81080442667007446, 0.016420882195234299]]}
#
```

```

# So you can write them out or test them:
print "The line-fit parameters were:"
print "      maximum = %6.3f +/- %6.3f K" %\
      (fit_stat['peak'][0][0],fit_stat['peak'][0][1])
print "      center = %6.1f +/- %6.1f channels" %\
      (fit_stat['cent'][0][0],fit_stat['cent'][0][1])
print "      FWHM = %6.2f +/- %6.2f channels" %\
      (fit_stat['fwhm'][0][0],fit_stat['fwhm'][0][1])
#
# Which gives:
# The line-fit parameters were:
#      maximum = 0.811 +/- 0.016 K
#      center = 4091.0 +/- 0.7 channels
#      FWHM = 72.90 +/- 1.70 channels

# We can do the fit in km/s also
specunit = 'km/s'
# For some reason we need to help it along with a mask
maskline = [-50,0]

fitfile = 'sdusecase_orions_hc3n_kms.fit'
fit_stat_kms = sdfit()
# Should give (if in verbose mode)
# 0: peak = 0.811 K , centre = -27.134 km/s, FWHM = 2.933 km/s
#      area = 2.531 K km/s
#

# with
fit_stat_kms
# giving
# {'cent': [[-27.133651733398438, 0.016480101272463799]],
#  'fwhm': [[2.93294358253479, 0.038807671517133713]],
#  'nfit': 1,
#  'peak': [[0.81080895662307739, 0.0092909494414925575]]}

print "The line-fit parameters were:"
print "      maximum = %6.3f +/- %6.3f K" %\
      (fit_stat_kms['peak'][0][0],fit_stat_kms['peak'][0][1])
print "      center = %6.2f +/- %6.2f km/s" %\
      (fit_stat_kms['cent'][0][0],fit_stat_kms['cent'][0][1])
print "      FWHM = %6.4f +/- %6.4f km/s" %\
      (fit_stat_kms['fwhm'][0][0],fit_stat_kms['fwhm'][0][1])

```

```
# The line-fit parameters were:
#     maximum = 0.811 +/- 0.009 K
#     center = -27.13 +/- 0.02 km/s
#     FWHM = 2.9329 +/- 0.0388 km/s
```

```
#####
#
# End ORION-S Use Case
#
#####
```

A.2.2.2 Imaging of Total Power Raster Scans

This example illustrates the use of `sdt pimaging` for the total power raster scans of the Moon taken at ATF.

```
# load single dish module
# asap_init()

# The data used here (uid__X1e1_X3197_X1.ms) is the total power
# raster scans of the Moon taken at ATF (with both antennas).
# It is in MS format which was converted from the ASDM format.

# Do data plotting only
default(sdt pimaging)
inp()
plotlevel=2
# select antenna 1 (Vertex antenna)
antenna='1'
sdfile='uid__X1e1_X3197_X1.ms'
sdt pimaging()

# Now, rerun sdt pimaging to do actual data reduction (applying
# baseline subtraction from each scan, and then do imaging).
#
# Do baseline subtraction
calmode='baseline'
masklist=[30] # use 30 data points from each end of scan for fitting
# Do imaging
createimage=True
imagenname='moon.im'
imagesize=[200,200]
cell=[0.2] # in arcmin
phasecenter='AZEL 187d54m22s 41d03m0s'
```

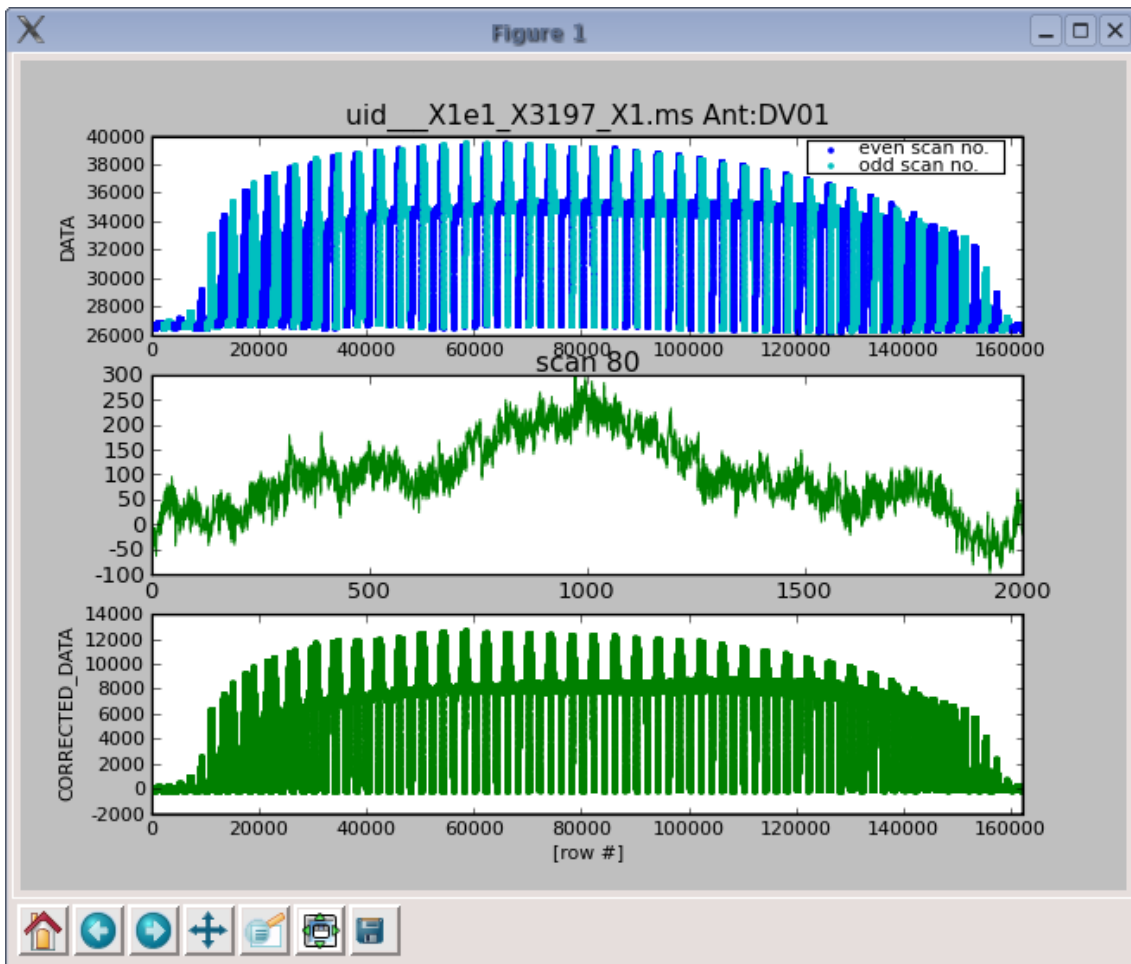


Figure A.2: Total power data display using `sdtipimaging`, with `calmode='baseline'`. The top panel shows uncalibrated data versus row numbers. The middle panel shows baseline fitting of each scan (only shown here the last scan). The bottom panel shows the calibrated (baseline subtracted) data.

```
ephemsrcname='moon' # specify ephemeris source name (can be omitted)
plotlevel=1
#plotlevel=2 to see progress of each fitting
sdtipimaging()
```

A.3 Using The ASAP Toolkit Within CASA

ASAP is included with the CASA installation/build. It is not loaded upon start-up, however, and must be imported as a standard Python package. A convenience function exists for importing

ASAP along with a set of prototype tasks for single dish analysis:

```
CASA <1>: asap_init
```

Once this is done, all of the ASAP functionality is now under the Python 'sd' tool. **Note:** This means that if you are following the ASAP cookbook or documentation, all of the commands should be invoked with a 'sd.' before the native ASAP command.

The ASAP interface is essentially the same as that of the CASA toolkit, that is, there are groups of functionality (aka tools) which have the ability to operate on your data. Type:

```
CASA <4>: sd.<TAB>
sd.__builtins__      sd._validate_bool    sd.list_scans
sd.__class__         sd._validate_int     sd.mask_and
sd.__date__          sd.asapfitter        sd.mask_not
sd.__delattr__       sd.asaplinefind     sd.mask_or
sd.__dict__          sd.asaplog           sd.merge
sd.__doc__           sd.asaplotbase      sd.os
sd.__file__          sd.asaplotgui        sd.plf
sd.__getattr__       sd.asapmath          sd.plotter
sd.__hash__          sd.asapplotter      sd.print_log
sd.__init__          sd.asapreader        sd.quotient
sd.__name__          sd.average_time      sd.rc
sd.__new__           sd.calfs             sd.rcParams
sd.__path__          sd.calnod            sd.rcParamsDefault
sd.__reduce__        sd.calps             sd.rc_params
sd.__reduce_ex__     sd.casapath          sd.rcdefaults
sd.__repr__          sd.commands          sd.reader
sd.__revision__      sd.defaultParams    sd.revinfo
sd.__setattr__       sd.dosigref          sd.scantable
sd.__str__           sd.dototalpower      sd.selector
sd.__version__       sd.fitter            sd.simple_math
sd._asap             sd.interactivemask   sd.sys
sd._asap_fname       sd.is_ipython         sd.unique
sd._asaplog          sd.linecatalog        sd.version
sd._is_sequence_or_number sd.linefinder        sd.welcome
sd._n_bools          sd.list_files        sd.xyplotter
```

...to see the list of tools.

In particular, the following are essential for most reduction sessions:

- `sd.scantable` - the data structure for ASAP and the core methods for manipulating the data; allows importing data, making data selections, basic operations (averaging, baselines, etc) and setting data characteristics (e.g., frequencies, etc).
- `sd.selector` - selects a subset of data for subsequent operations
- `sd.fitter` - fit data

- `sd.plotter` - plotting facilities (uses `matplotlib`)

The `scantable` functions are used most often and can be applied to both the initial scantable and to any spectrum from that scan table. Type

```
sd.scantable.<TAB>
```

(using TAB completion) to see the full list.

A.3.1 Environment Variables

The `asaprc` environment variables are stored in the Python dictionary `sd.rcParams` in CASA. This contains a number of parameters that control how ASAP runs, for both tools and tasks. You can see what these are set to by typing at the CASA prompt:

```
CASA <2>: sd.rcParams
Out[2]:
{'insitu': True,
 'plotter.colours': '',
 'plotter.decimate': False,
 'plotter.ganged': True,
 'plotter.gui': True,
 'plotter.histogram': False,
 'plotter.linestyles': '',
 'plotter.panelling': 's',
 'plotter.papertype': 'A4',
 'plotter.stacking': 'p',
 'scantable.autoaverage': True,
 'scantable.freqframe': 'LSRK',
 'scantable.save': 'ASAP',
 'scantable.storage': 'memory',
 'scantable.verbosesummary': False,
 'useplotter': True,
 'verbose': True}
```

The use of these parameters is described in detail in the ASAP Users Guide.

These parameters can be changed through the `sd.rc` function. Use is described in `help sd.rc`:

```
CASA <3>: help(sd.rc)
Help on function rc in module asap:

rc(group, **kwargs)
    Set the current rc params. Group is the grouping for the rc, eg
    for scantable.save the group is 'scantable', for plotter.stacking, the
    group is 'plotter', and so on. kwargs is a list of attribute
    name/value pairs, eg
```

```
rc('scantable', save='SDFITS')
```

sets the current rc params and is equivalent to

```
rcParams['scantable.save'] = 'SDFITS'
```

Use `rcdefaults` to restore the default rc params after changes.

A.3.2 Import

Data can be loaded into ASAP by using the `scantable` function which will read a variety of recognized formats (RPFITS, varieties of SDFITS, the CASA Measurement Set, and NRO OTF data format). For example:

```
CASA <1>: scans = sd.scantable('OrionS_rawACSmod', average=False)
Importing OrionS_rawACSmod...
```

A.3.2.1 General descriptions

The following are some cautions when using this import feature.

- It is important to specify the `antenna` parameter in the Measurement Set conversion. For example:

```
CASA<1>: scans = sd.scantable( 'OrionS_rawACSmod', average=False, antenna=0 )
```

The value of the `antenna` parameter can be either `id` (integer) or `name` (string). The default value for `antenna` parameter is 0.

- It is important to use the `average=False` parameter setting as the calibration routines supporting GBT data require all of the individual times and phases.
- GBT data may need some pre-processing prior to using ASAP. In particular, the program which converts GBT raw data into CASA Measurement Sets tends to proliferate the number of spectral windows due to shifts in the tracking frequency; this is being worked on by GBT staff. In addition, GBT SDFITS is currently not readable by ASAP (in progress).
- Identification of observing modes or reference and source data are changed from the name assignment to the identification number assignment. Position switched (`'_ps'`), Nod (`'_nod'`), and frequency switched (`'_fs'`) data are assigned their id number as 0, 2, and 3, respectively. The corresponding reference data (name with `'r'`) for position switched and frequency switched modes are assigned as 1 and 4, respectively. These identification number is stored as `SRCTYPE` in the scantable.
- Importing of Nobeyama Radio Observatory (NRO) data (in both OTF and NEWSTAR format) is now available. However, it is still experimental and only tested to work from toolkit level.

A.3.2.2 Handling ALMA data

- Currently ASDM data cannot be read to ASAP directly. It must be converted to a MS first via `importasdm` task. Then you can load this MS to ASAP. The `importasdm` task allows to handle these two steps at once. To do that, you should set `singledish=True` and specify id or name of the antenna by the `antenna` parameter (see 2.2.1).
- If the MS data contain data from multiple single dish antennas you need either, to specify the `antenna` parameter when importing data (`sd.scantable`) or, to split the data by antenna using `sd.splitant` for further processing in ASAP since ASAP `scantable` cannot properly store the data from multiple antennas. The method `sd.splitant` splits a measurement set by antenna ID and save the tables as scantables containing data from each antenna. The names of output scantables are defined as prefix specified by users (with parameter `outprefix`) + '.' + antenna name. For example, if you split measurement set, `foo.ms`, which contains data from `antA` and `antB`, with `outprefix='splitted'`, i.e., `sd.splitant('foo.ms', outprefix='splitted')`, the names of scantables are `'splitted.antA'` and `'splitted.antB'`. The returned value of the method is a list of scantable names.
- It is possible to read the 2-element interferometric data taken by the ALMA telescopes as a single dish data to further examine it using ASAP Toolkit and the SDtasks. The usage is the same as importing single dish data. For example, to load the ALMA interferometry data, `myOSFint.ms`

```
s=sd.scantable('myOSFint.ms', 'False')
```

It is still experimental and limited to data obtained with a two-element array.

Use the `summary` function to examine the data and get basic information:

```
CASA <8>: scans.summary()
```

```
-----
Scan Table Summary
-----
Beams:          1
IFs:           26
Polarisations: 2 (linear)
Channels:      8192

Observer:      Joseph McMullin
Obs Date:     2006/01/19/01:45:58
Project:      AGBT06A_018_01
Obs. Type:    OffOn:PSWITCHOFF:TPWCAL
Antenna Name: GBT
Flux Unit:    Jy
Rest Freqs:   [4.5490258e+10] [Hz]
Abcissa:      Channel
Selection:    none
```

Scan	Source	Time	Integration		
Beam	Position (J2000)				
IF	Frame	RefVal	RefPix	Increment	
20	OrionS_psr	01:45:58	4 x	30.0s	
0	05:15:13.5	-05.24.08.2			
	0	LSRK	4.5489354e+10	4096	6104.233
	1	LSRK	4.5300785e+10	4096	6104.233
	2	LSRK	4.4074929e+10	4096	6104.233
	3	LSRK	4.4166215e+10	4096	6104.233
21	OrionS_ps	01:48:38	4 x	30.0s	
0	05:35:13.5	-05.24.08.2			
	0	LSRK	4.5489354e+10	4096	6104.233
	1	LSRK	4.5300785e+10	4096	6104.233
	2	LSRK	4.4074929e+10	4096	6104.233
	3	LSRK	4.4166215e+10	4096	6104.233
22	OrionS_psr	01:51:21	4 x	30.0s	
0	05:15:13.5	-05.24.08.2			
	0	LSRK	4.5489354e+10	4096	6104.233
	1	LSRK	4.5300785e+10	4096	6104.233
	2	LSRK	4.4074929e+10	4096	6104.233
	3	LSRK	4.4166215e+10	4096	6104.233
23	OrionS_ps	01:54:01	4 x	30.0s	
0	05:35:13.5	-05.24.08.2			
	0	LSRK	4.5489354e+10	4096	6104.233
	1	LSRK	4.5300785e+10	4096	6104.233
	2	LSRK	4.4074929e+10	4096	6104.233
	3	LSRK	4.4166215e+10	4096	6104.233
24	OrionS_psr	02:01:47	4 x	30.0s	
0	05:15:13.5	-05.24.08.2			
	12	LSRK	4.3962126e+10	4096	6104.2336
	13	LSRK	4.264542e+10	4096	6104.2336
	14	LSRK	4.159498e+10	4096	6104.2336
	15	LSRK	4.3422823e+10	4096	6104.2336
25	OrionS_ps	02:04:27	4 x	30.0s	
0	05:35:13.5	-05.24.08.2			
	12	LSRK	4.3962126e+10	4096	6104.2336
	13	LSRK	4.264542e+10	4096	6104.2336
	14	LSRK	4.159498e+10	4096	6104.2336
	15	LSRK	4.3422823e+10	4096	6104.2336
26	OrionS_psr	02:07:10	4 x	30.0s	
0	05:15:13.5	-05.24.08.2			
	12	LSRK	4.3962126e+10	4096	6104.2336
	13	LSRK	4.264542e+10	4096	6104.2336
	14	LSRK	4.159498e+10	4096	6104.2336
	15	LSRK	4.3422823e+10	4096	6104.2336
27	OrionS_ps	02:09:51	4 x	30.0s	
0	05:35:13.5	-05.24.08.2			
	12	LSRK	4.3962126e+10	4096	6104.2336
	13	LSRK	4.264542e+10	4096	6104.2336
	14	LSRK	4.159498e+10	4096	6104.2336

```
15      LSRK    4.3422823e+10   4096   6104.2336
```

A.3.3 Scantable Manipulation

Within ASAP, data is stored in a `scantable`, which holds all of the observational information and provides functionality to manipulate the data and information. The building block of a `scantable` is an integration which is a single row of a `scantable`. Each row contains just one spectrum for each beam, IF and polarization.

Once you have a `scantable` in ASAP, you can select a subset of the data based on scan numbers, sources, or types of scan; note that each of these selections returns a new 'scantable' with all of the underlying functionality:

```
CASA <5>: scan27=scans.get_scan(27)           # Get the 27th scan
CASA <6>: scans20to24=scans.get_scan(range(20,25)) # Get scans 20 - 24
CASA <7>: scans_on=scans.get_scan('*_ps')      # Get ps scans on source
CASA <8>: scansOrion=scans.get_scan('Ori*')    # Get all Orion scans
```

To copy a `scantable`, do:

```
CASA <15>: ss=scans.copy()
```

A.3.3.1 Data Selection

In addition to the basic data selection above, data can be selected based on IF, beam, polarization, scan number as well as values such as `Tsys`. To make a selection create a `selector` object choose among various selection functions, e.g.,

```
sel = sd.selector()           # initialize a selector object
                                # sel.<TAB> will list all options
sel.set_ifs(0)                # select only the first IF of the data
scans.set_selection(sel)     # apply the selection to the data
print scans                   # shows just the first IF
```

A.3.3.2 State Information

Some properties of a `scantable` apply to all of the data, such as example, spectral units, frequency frame, or Doppler type. This information can be set using the `scantable` `_set_xxxx_` methods. These are currently:

```
CASA <1>: sd.scantable.set_<TAB>
sd.scantable.set_dirframe      sd.scantable.set_fluxunit      sd.scantable.set_restfreqs
sd.scantable.set_doppler       sd.scantable.set_freqframe     sd.scantable.set_selection
sd.scantable.set_feedtype      sd.scantable.set_instrument    sd.scantable.set_unit
```

For example, `sd.scantable.set_fluxunit` sets the default units that describe the flux axis:

```
scans.set_fluxunit('K') # Set the flux unit for data to Kelvin
```

Choices are 'K' or 'Jy'. Note: the `scantable.set_fluxunit` function only changes the **name** of the current fluxunit. To change fluxunits, use `scantable.convert_flux` as described in § A.3.4.2 instead (currently it is necessary to do some gymnastics for non-AT telescopes).

Use `sd.scantable.set_unit` to set the units to be used on the spectral axis:

```
scans.set_unit('GHz') # Use GHz as the spectral axis for plots
```

The choices for the units are 'km/s', 'channel', or '*Hz' (e.g. 'GHz', 'MHz', 'kHz', 'Hz'). This does the proper conversion using the current frame and Doppler reference as can be seen when the spectrum is plotted.

Set the frame in which the frequency (spectral) axis is defined by `sd.scantable.set_freqframe`:

```
CASA <2>: help(sd.scantable.set_freqframe)
Help on method set_freqframe in module asap.scantable:
```

```
set_freqframe(self, frame=None) unbound asap.scantable.scantable method
  Set the frame type of the Spectral Axis.
  Parameters:
    frame:  an optional frame type, default 'LSRK'. Valid frames are:
            'REST', 'TOPO', 'LSRD', 'LSRK', 'BARY',
            'GEO', 'GALACTO', 'LGROUP', 'CMB'
  Examples:
    scan.set_freqframe('BARY')
```

The most useful choices here are `frame = 'LSRK'` (the default for the function) and `frame = 'TOPO'` (what the GBT actually observes in). Note that the 'REST' option is not yet available. The Doppler frame is set with `sd.scantable.set_doppler`:

```
CASA <3>: help(sd.scantable.set_doppler)
Help on method set_doppler in module asap.scantable:
```

```
set_doppler(self, doppler='RADIO') unbound asap.scantable.scantable method
  Set the doppler for all following operations on this scantable.
  Parameters:
    doppler:  One of 'RADIO', 'OPTICAL', 'Z', 'BETA', 'GAMMA'
```

Finally, there are a number of functions to query the state of the scantable. These can be found in the usual way:

```
CASA <4>: sd.scantable.get<TAB>
sd.scantable.get_abcissa      sd.scantable.get_restfreqs      sd.scantable.getbeamnos
sd.scantable.get_azimuth     sd.scantable.get_scan           sd.scantable.getcycle
sd.scantable.get_column_names sd.scantable.get_selection     sd.scantable.getif
```

```

sd.scantable.get_direction      sd.scantable.get_sourcename    sd.scantable.getifnos
sd.scantable.get_elevation     sd.scantable.get_time         sd.scantable.getpol
sd.scantable.get_fit           sd.scantable.get_tsys         sd.scantable.getpolnos
sd.scantable.get_fluxunit      sd.scantable.get_unit         sd.scantable.getscan
sd.scantable.get_parangle      sd.scantable.getbeam          sd.scantable.getscannos

```

These include functions to get the current values of the states mentioned above, as well as methods to query the number of scans, IFs, and polarizations in the scantable and their designations. See the inline help of the individual functions for more information.

A.3.3.3 Masks

Several functions (fitting, baseline subtraction, statistics, etc) may be run on a range of channels (or velocity/frequency ranges). You can create masks of this type using the `create_mask` function:

```

# spave = an averaged spectrum
spave.set_unit('channel')
rmsmask=spave.create_mask([5000,7000]) # create a region over channels 5000-7000
rms=spave.stats(stat='rms',mask=rmsmask) # get rms of line free region

rmsmask=spave.create_mask([3000,4000],invert=True) # choose the region
# *excluding* the specified channels

```

The mask is stored in a simple Python variable (a list) and so may be manipulated using an Python facilities.

A.3.3.4 Scantable Management

scantables can be listed via:

```

CASA <33>: sd.list_scans()
The user created scantables are:
['scans20to24', 's', 'scan27']

```

As every `scantable` will consume memory, if you will not use it any longer, you can explicitly remove it via:

```

del <scantable name>

```


A.3.3.5 Scantable Mathematics

It is possible to do simple mathematics directly on `scantables` from the CASA command line using the `+`, `-`, `*`, `/` operators as well as their cousins `+=`, `-=`, `*=`, `/=`.

```
CASA <10>: scan2=scan1+2.0 # add 2.0 to data
CASA <11>: scan *= 1.05    # scale spectrum by 1.05
```

Operands can be a numerical value and one- or two-dimensional Python list. For list operand, it should conform with the shape of the scantable. Mathematics between two scantables is also available. In that case, scantables must conform with each other.

NOTE: In scantable mathematics, scantable must be put on the left. For example:

```
CASA<12>: scan2=scan1+2.0    # this works
CASA<13>: scan2=2.0+scan1    # this causes an error
```

A.3.3.6 Scantable Save and Export

ASAP can save scantables in a variety of formats, suitable for reading into other packages. The formats are:

- **ASAP** – This is the internal format used for ASAP. It is the only format that allows the user to restore the data, fits, etc., without losing any information. As mentioned before, the ASAP scantable is a CASA Table (memory-based table). This function just converts it to a disk-based table. You can access it with the CASA `browsetable` task or any other CASA table tasks.
- **SDFITS** – The Single Dish FITS format. This format was designed for interchange between packages but few packages can actually read it.
- **ASCII** – A simple text based format suitable for the user to process using Python or other means.
- **Measurement Set (V2: CASA format)** – Saves the data in a Measurement Set. All CASA tasks which use an MS should work with this format.

```
scans.save('output_filename','format'), e.g.,
CASA <19>: scans.save('FLS3a_calfs','MS2')
```

A.3.4 Calibration

For some observatories, the calibration happens transparently as the input data contains the `Tsys` measurements taken during the observations. The nominal `'Tsys'` values may be in Kelvin or Jansky. The user may wish to apply a `Tsys` correction or apply gain-elevation and opacity corrections.

A.3.4.1 Tsys scaling

If the nominal Tsys measurement at the telescope is wrong due to incorrect calibration, the `scale` function allows it to be corrected.

```
scans.scale(1.05,tsys=True) # by default only the spectra are scaled
                           # (and not the corresponding Tsys) unless tsys=True
```

A.3.4.2 Flux and Temperature Unit Conversion

To convert measurements in Kelvin to Jansky (and vice versa), the `convert_flux` function may be used. This converts and scales the data to the selected units. The user may need to supply the aperture efficiency, telescope diameter, or the Jy/K factor

```
scans.convert_flux(eta=0.48, d=35.) # Unknown telescope
scans.convert_flux(jypk=15) # Unknown telescope (alternative)
scans.convert_flux() # known telescope (mostly AT telescopes)
scans.convert_flux(eta=0.48) # if telescope diameter known
```

A.3.4.3 Gain-Elevation and Atmospheric Optical Depth Corrections

At higher frequencies, it is important to make corrections for atmospheric opacity and gain-elevation effects. **NOTE:** Currently, the MS to scantable conversion does not adequately populate the azimuth and elevation in the `scantable`. As a result, one must calculate these via:

```
scans.recalc_azel()
Computed azimuth/elevation using
Position: [882590, -4.92487e+06, 3.94373e+06]
Time: 01:48:38 Direction: 05:35:13.5 -05.24.08.2
=> azel: 154.696 43.1847 (deg)
Time: 01:48:38 Direction: 05:35:13.5 -05.24.08.2
=> azel: 154.696 43.1847 (deg)
Time: 01:48:38 Direction: 05:35:13.5 -05.24.08.2
=> azel: 154.696 43.1847 (deg)
Time: 01:48:38 Direction: 05:35:13.5 -05.24.08.2
=> azel: 154.696 43.1847 (deg)
Time: 01:48:38 Direction: 05:35:13.5 -05.24.08.2
=> azel: 154.696 43.1847 (deg)
...
```

With the correct Az/El, it can be corrected for a *known* opacity by:

```
scans.opacity(tau=0.09) # Opacity from which the correction factor:
                       # exp(tau*zenith-distance)
```

A.3.4.4 Calibration of GBT data

Data from the GBT are uncalibrated and come as sets of integrations representing the different phases of a calibration cycle (e.g., on source, calibration on, on source, calibration off, on reference, calibration on; on reference, calibration off). Currently, there are a number of routines emulating the standard GBT calibration (in GBTIDL):

- `calps` - calibrate position switched data
- `calfs` - calibrate frequency switched data
- `calnod` - calibration nod (beam switch) data

All these routines calibrate the spectral data to antenna temperature adopting the GBT calibration method as described in the GBTIDL calibration document available at:

- http://wwwlocal.gb.nrao.edu/GBT/DA/gbtidl/gbtidl_calibration.pdf

There are two basic steps:

First: determine system temperature using a noise tube calibrator (`sd.dototalpower()`)

For each integration, the system temperature is calculated from CAL noise on/off data as:

$$T_{sys} = T_{cal} \times \frac{\langle ref_{caloff} \rangle}{\langle ref_{calon} - ref_{caloff} \rangle} + \frac{T_{cal}}{2}$$

`ref` which refers to reference data and the spectral data are averaged across the bandpass. Note that the central 80% of the spectra are used for the calculation.

Second, determine antenna temperature (`sd.dosigref()`)

The antenna temperature for each channel is calculated as:

$$T_a(\nu) = T_{sys} \times \frac{sig(\nu) - ref(\nu)}{ref(\nu)}$$

where $sig = \frac{1}{2}(sig_{calon} + sig_{caloff})$, $ref = \frac{1}{2}(sig_{calon} + sig_{caloff})$.

Each calibration routine may be used as:

```
scans=sd.scantable('inputdata',False)           # create a scantable called 'scans'
calibrated_scans = sd.calps(scans,[scanlist])   # calibrate scantable with position-switched
                                                # scheme
```

Note: For `calps` and `calnod`, the `scanlist` must be scan pairs in correct order as these routines only do minimal checking.

A.3.4.5 Comprehensive calibration function

A new function `calibrate` is introduced for more comprehensive calibration. This function calls appropriate calibration scheme by referring the antenna name in the metadata. The user just specify input scantable and `calmode` parameter that indicates calibration mode ('ps', 'nod', 'fs', or 'otf'). For GBT data, `calibrate` function calls one of the functions listed in the above section depending upon the value of `calmode`. For APEX data, `apexcal` function is called, while for ALMA data, that contains string 'ALMA' or 'OSF' in its antenna name, `almacal` function is called. Calibration scheme for these two telescopes are essentially same, and can be expressed as,

$$T_a^* = T_{\text{sys}} * (\text{ON} - \text{OFF}) / \text{OFF},$$

where OFF scans are interpolated in time if possible.

A.3.5 Averaging

One can average polarizations in a scantable using the `sd.scantable.average_pol` function:

```
averaged_scan = scans.average_pol(mask,weight)
```

where:

Parameters:

`mask`: An optional mask defining the region, where the averaging will be applied. The output will have all specified points masked.

`weight`: Weighting scheme. 'none' (default), 'var' (1/var(spec) weighted), or 'tsys' (1/Tsys**2 weighted)

Example:

```
spave = stave.average_pol(weight='tsys')
```

One can also average scans over time using `sd.average_time`:

```
sd.average_time(scantable,mask,scanav,weight,align)
```

where:

Parameters:

one scan or comma separated scans

`compel`: if True, enable averaging of multi-resolution spectra.

`mask`: an optional mask (only used for 'var' and 'tsys' weighting)

`scanav`: True averages each scan separately.
False (default) averages all scans together,

`weight`: Weighting scheme.
'none' (mean no weight)

```

    'var'      (1/var(spec) weighted)
    'tsys'     (1/Tsys**2 weighted)
    'tint'     (integration time weighted)
    'tintsys'  (Tint/Tsys**2)
    'median'   ( median averaging)
align:  align the spectra in velocity before averaging. It takes
        the time of the first spectrum in the first scantable
        as reference time.

```

Example:

```
stave = sd.average_time(scans,weight='tintsys')
```

Note that alignment of the velocity frame should be done before averaging if the time spanned by the scantable is long enough. This is done through the `align=True` option in `sd.average_time`, or explicitly through the `sd.scantable.freq_align` function, e.g.,

```

CASA <62>: sc = sd.scantable('orions_scan20to23_if0to3.asap',False)
CASA <63>: sc.freq_align()
Aligned at reference Epoch 2006/01/19/01:49:23 (UTC) in frame LSRK
CASA <64>: av = sd.average_times(sc)

```

The time averaging can also be applied to multiple scantables. For example, such data might have been taken on different days. The `sd.average_time` function takes multiple scantables as input. However, if they were taken at significantly different times (different days for example), then `sd.scantable.freq_align` must be used to align the velocity scales to the same time, e.g.

```

CASA <65>: sc1 = sd.scantable('orions_scan21_if0to3.asap',False)
CASA <66>: sc2 = sd.scantable('orions_scan23_if0to3.asap',False)
CASA <67>: sc1.freq_align()
Aligned at reference Epoch 2006/01/19/01:49:23 (UTC) in frame LSRK
CASA <68>: sc2.freq_align(reftime='2006/01/19/01:49:23')
Aligned at reference Epoch 2006/01/19/01:54:46 (UTC) in frame LSRK
CASA <69>: scav = sd.average_times(sc1,sc2)

```

A.3.6 Spectral Smoothing

Smoothing on data can be done as follows:

```

scantable.smooth(kernel,      # type of smoothing: 'hanning' (default), 'gaussian', 'boxcar'
                 width,      # width in pixls (ignored for hanning); FWHM for gaussian.
                 insitu)     # if False (default), do smoothing in-situ; otherwise,
                             # make new scantable

```

Example:

```

# spave is an averaged spectrum
spave.smooth('boxcar',5)    # do a 5 pixel boxcar smooth on the spectrum
sd.plotter.plot(spave)     # should see smoothed spectrum

```

A.3.7 Baseline Fitting

The function `sd.scantable.poly_baseline` carries out a baseline fit, given an mask of channels (if desired):

```
msk=scans.create_mask([100,400],[600,900])
scans.poly_baseline(msk,order=1)
```

This will fit a first order polynomial to the selected channels and subtract this polynomial from the full spectrum.

The `auto_poly_baseline` function can be used to automatically baseline your data without specifying channel ranges for the line-free data. It automatically figures out the line-free emission and fits a polynomial baseline to that data. The user can use masks to fix the range of channels or velocity range for the fit as well as mark the band edge as invalid:

```
scans.auto_poly_baseline(mask,edge,order,threshold,chan_avg_limit,plot,insitu):
```

Parameters:

`mask`: an optional mask retrieved from `scantable`

`edge`: an optional number of channel to drop at the edge of spectrum. If only one value is specified, the same number will be dropped from both sides of the spectrum. Default is to keep all channels. Nested tuples represent individual edge selection for different IFs (a number of spectral channels can be different)

`order`: the order of the polynomial (default is 0)

`threshold`: the threshold used by line finder. It is better to keep it large as only strong lines affect the baseline solution.

`chan_avg_limit`: the maximum number of consecutive spectral channels to average during the search of weak and broad lines. The default is no averaging (and no search for weak lines). If such lines can affect the fitted baseline (e.g. a high order polynomial is fitted), increase this parameter (usually values up to 8 are reasonable). Most users of this method should find the default value sufficient.

`plot`: plot the fit and the residual. In this each individual fit has to be approved, by typing 'y' or 'n'

`insitu`: if False a new `scantable` is returned. Otherwise, the scaling is done in-situ. The default is taken from `.asaprc` (False)

Example:

```
scans.auto_poly_baseline(order=2,threshold=5)
```

A.3.8 Line Fitting

Multi-component Gaussian fitting is done by creating a fitting object, specifying fit parameters and finally fitting the data. Fitting can be done on a `scantable` selection or an entire `scantable` using the `auto_fit` function.

```
#spave is an averaged spectrum
f=sd.fitter()                # create fitter object
msk=spave.create_mask([3928,4255]) # create mask region around line
f.set_function(gauss=1)     # set a single gaussian component
f.set_scan(spave,msk)      # set the scantable and region
#
# Automatically guess start values
f.fit()                     # fit
f.plot(residual=True)      # plot residual
f.get_parameters()         # retrieve fit parameters
# 0: peak = 0.786 K , centre = 4091.236 channel, FWHM = 70.586 channel
#    area = 59.473 K channel
f.store_fit('orions_hc3n_fit.txt') # store fit
#
# To specify an initial guess:
f.set_function(gauss=1)    # set a single gaussian component
f.set_gauss_parameters(0.4,4100,200\ # set initial guesses for Gaussian
    ,component=0)         # for first component (0)
# (peak,center,fwhm)
#
# For multiple components set
# initial guesses for each, e.g.,
f.set_function(gauss=2)   # set two gaussian components
f.set_gauss_parameters(0.4,4100,200\ # set initial guesses for Gaussian
    ,component=0)         # for first component (0)
f.set_gauss_parameters(0.1,4200,100\ # set initial guesses for Gaussian
    ,component=1)         # for second component (1)
```

A.3.9 Plotting

A.3.9.1 ASAP plotter

The ASAP plotter uses the same Python matplotlib library as in CASA (for x-y plots). It is accessed via the:

```
sd.plotter<TAB> # see all functions (omitted here)
sd.plotter.plot(scans) # the workhorse function
sd.plotter.set<TAB>
sd.plotter.set_abscissa      sd.plotter.set_layout      sd.plotter.set_panelling
sd.plotter.set_colors       sd.plotter.set_legend     sd.plotter.set_range
sd.plotter.set_colours     sd.plotter.set_linestyles sd.plotter.set_selection
```

```
sd.plotter.set_data      sd.plotter.set_mask      sd.plotter.set_stacking
sd.plotter.set_font     sd.plotter.set_mode     sd.plotter.set_title
```

Spectra can be plotted at any time when `refresh = True` (default) is selected, and it will attempt to do the correct layout depending on whether it is a set of scans or a single scan. You can switch off verbose plotting by `refresh = False` in tool parameters for faster plotting in scripts:

```
sd.plotter.set_data(scan,refresh=False) # set scantable to plot
                                         # this should be done at first.
sd.plotter.set_mode(stacking='time',panelling='if',refresh=False)
sd.plotter.set_range(ystart=-1.0,yend=5.0,refresh=False)
sd.plotter.plot()                       # actual plotting
```

The details of the plotter display (matplotlib) are detailed in the earlier section.

A.3.9.2 Line Catalog

ASAP allows loading a custom line catalog in ASCII format. The ASCII text file must have at least 4 columns with Molecule name, frequency in MHz, frequency error and intensity (any units). If the molecule name contains any spaces, they must be wrapped in quotes `”`. A sample of the ASCII catalog is shown below.

```
H2D+    3955.2551 228.8818 -7.1941
H2D+    12104.7712 177.1558 -6.0769
H2D+    45809.2731 118.3223 -3.9494
CH       701.6811   .0441 -7.1641
CH       724.7709   .0456 -7.3912
CH      3263.7940   .1000 -6.3501
CH      3335.4810   .1000 -6.0304
```

You can load the ASCII line catalog, for example, if it is called `my_custom_linecat.txt`, by following command.

```
mycatlog = sd.linecatlog('my_custom_linecat.txt')
```

Use `sd.plotter.plot_line` to overlay the line catalog on the plot. (Currently overplotting line catalog works only spectra plotted in frequency.)

```
scans.set_unit('GHz')
sd.plotter.plot(scans)
sd.plotter.plot_line(mycatlog)
```

The following are some useful functions to control the line catalog access. See ASAP User Guide for more complete descriptions.


```
mycatlog.save('my_custom_linecat.tbl') # save to the internal table format
mycatlog.set_frequency_limits(100,115,'GHz') #set a frequency range for line selection
mycatlog.set_name('*OH') # select all alchols
```

A.3.10 Setting/Getting Rest Frequencies

The rest frequencies used in the data can be retrieved by `sd.scantable.get_restfreqs()` and set to new values by `sd.scantable.set_restfreqs()`. The CASA version of ASAP now can store multiple rest frequencies for each IF.

```
scans.get_restfreqs()          #retrieve current rest frequencies
#{0: [45490258000.0]}
```

All of the rest frequencies currently set to the data are listed in python dictionary for each MOLECULE.ID.

Here is an example of setting multiple rest frequencies for spectra of a particular IF:

```
#Select IFs, then set rest frequencies,
sel=sd.selector()
sel.setifs(0)
scans.set_selection(sel)
scans.set_restfreqs([45490258000.0,45590258000.0,45690258000.0])
```

NOTE: there is no functionality yet to select a specific rest frequency to apply to a specific line, etc. Currently, the first one in the list of the rest frequencies is used for such calculation.

A.3.11 Single Dish Spectral Analysis Use Case With ASAP Toolkit

Below is a script that illustrates how to reduce single dish data using ASAP within CASA. First a summary of the dataset is given and then the script.

```
#           MeasurementSet Name: /home/rohir3/jocular/SD/OrionS_rawACSmod      MS Version 2
#
# Project: AGBT06A_018_01
# Observation: GBT(1 antennas)
#
#Data records: 256           Total integration time = 1523.13 seconds
#  Observed from  01:45:58   to   02:11:21
#
#Fields: 4
#  ID  Name           Right Ascension  Declination  Epoch
#  0   OrionS         05:15:13.45     -05.24.08.20 J2000
#  1   OrionS         05:35:13.45     -05.24.08.20 J2000
#  2   OrionS         05:15:13.45     -05.24.08.20 J2000
#  3   OrionS         05:35:13.45     -05.24.08.20 J2000
#
#Spectral Windows: (8 unique spectral windows and 1 unique polarization setups)
```

```

# SpwID #Chans Frame Ch1(MHz) Resoln(kHz) TotBW(kHz) Ref(MHz) Corrs
# 0      8192 LSRK 45464.3506 6.10423298 50005.8766 45489.3536 RR LL HC3N
# 1      8192 LSRK 45275.7825 6.10423298 50005.8766 45300.7854 RR LL HN15CO
# 2      8192 LSRK 44049.9264 6.10423298 50005.8766 44074.9293 RR LL CH3OH
# 3      8192 LSRK 44141.2121 6.10423298 50005.8766 44166.2151 RR LL HCCC15N
# 12     8192 LSRK 43937.1232 6.10423356 50005.8813 43962.1261 RR LL HNCO
# 13     8192 LSRK 42620.4173 6.10423356 50005.8813 42645.4203 RR LL H15NCO
# 14     8192 LSRK 41569.9768 6.10423356 50005.8813 41594.9797 RR LL HNC180
# 15     8192 LSRK 43397.8198 6.10423356 50005.8813 43422.8227 RR LL SiO

```

```
# Scans: 21-24 Setup 1 HC3N et al
```

```
# Scans: 25-28 Setup 2 SiO et al
```

```
casapath=os.environ['AIPSPATH']
```

```

#ASAP script                                     # COMMENTS
#-----
import asap as sd                                #import ASAP package into CASA
                                                #Orion-S (SiO line reduction only)
                                                #Notes:
                                                #scan numbers (zero-based) as compared toGBTIDL

                                                #changes made to get to OrionS_rawACSmod
                                                #modifications to label sig/ref positions
os.environ['AIPSPATH']=casapath                 #set this environment variable back - ASAP changes it

s=sd.scantable('OrionS_rawACSmod',False)#load the data without averaging

s.summary()                                     #summary info
s.set_fluxunit('K')                             # make 'K' default unit
scal=sd.calps(s,[20,21,22,23])                 # Calibrate HC3N scans

scal.recalc_azel()                             # recalculate az/el to
scal.opacity(0.09)                             # do opacity correction
sel=sd.selector()                               # Prepare a selection
sel.set_ifs(0)                                  # select HC3N IF
scal.set_selection(sel)                         # get this IF
stave=sd.average_time(scal,weight='tintsys')   # average in time
spave=stave.average_pol(weight='tsys')         # average polarizations;Tsys-weighted (1/Tsys**2) average
sd.plotter.plot(spave)                         # plot

spave.smooth('boxcar',5)                       # boxcar 5
spave.auto_poly_baseline(order=2)              # baseline fit order=2
sd.plotter.plot(spave)                         # plot

spave.set_unit('GHz')
sd.plotter.plot(spave)

```

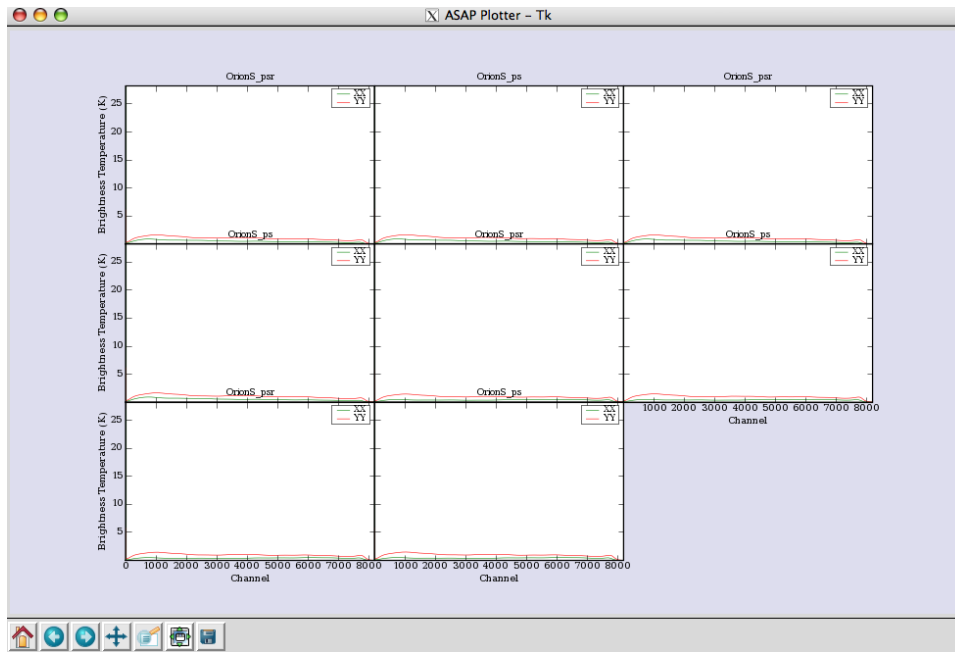


Figure A.3: Multi-panel display of the scantable. There are two plots per scan indicating the `_psr` (reference position data) and the `_ps` (source data).

```
sd.plotter.set_histogram(hist=True)      # draw spectrum using histogram
sd.plotter.axhline(color='r',linewidth=2) # zline
sd.plotter.save('orions_hc3n_reduced.eps') # save postscript spectrum

spave.set_unit('channel')
rmsmask=spave.create_mask([5000,7000]) # get rms of line free regions
rms=spave.stats(stat='rms',mask=rmsmask) # rms
#-----
#Scan[0] (OrionS_ps) Time[2006/01/19/01:52:05]:
# IF[0] = 0.048
#-----
# LINE

linemask=spave.create_mask([3900,4200])
max=spave.stats('max',linemask)      # IF[0] = 0.918
sum=spave.stats('sum',linemask)      # IF[0] = 64.994
median=spave.stats('median',linemask) # IF[0] = 0.091
mean=spave.stats('mean',linemask)    # IF[0] = 0.210

# Fitting
spave.set_unit('channel')             # set units to channel
sd.plotter.plot(spave)                 # plot spectrum
```

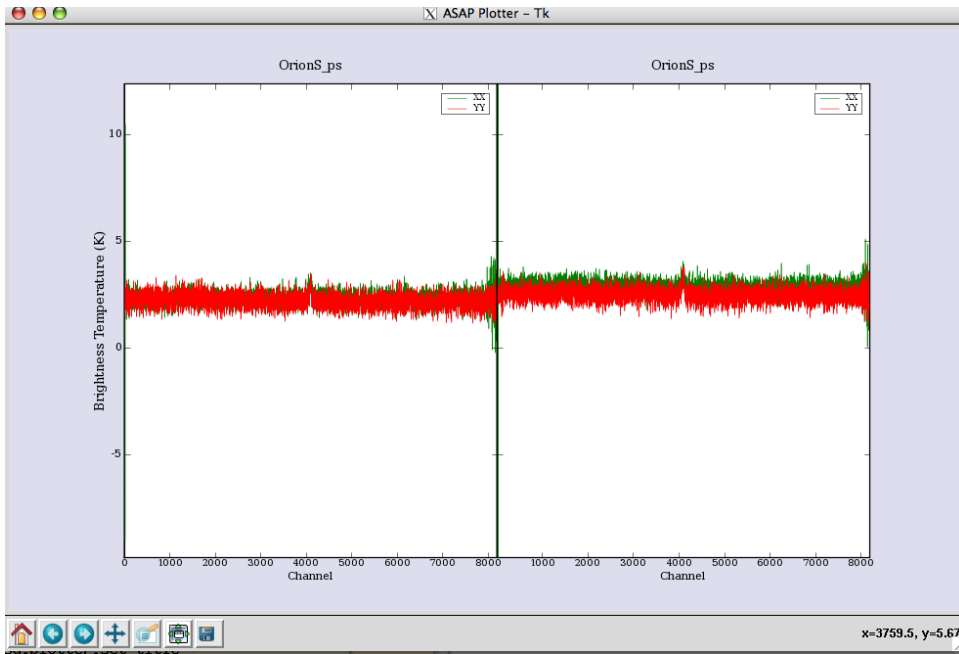


Figure A.4: Two panel plot of the calibrated spectra. The GBT data have a separate scan for the SOURCE and REFERENCE positions so scans 20,21,22 and 23 result in these two spectra.

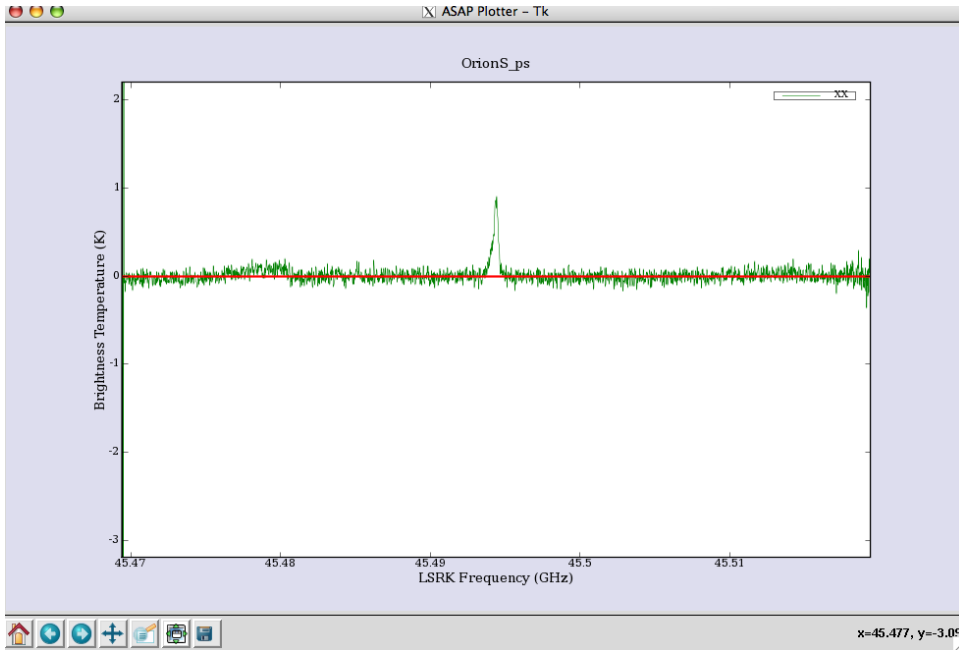


Figure A.5: Calibrated spectrum with a line at zero (using histograms).

```

f=sd.fitter()
msk=spave.create_mask([3928,4255])      # create region around line
f.set_function(gauss=1)                 # set a single gaussian component
f.set_scan(spave,msk)                   # set the data and region for the fitter
f.fit()                                  # fit
f.plot(residual=True)                   # plot residual

f.get_parameters()                       # retrieve fit parameters
# 0: peak = 0.786 K , centre = 4091.236 channel, FWHM = 70.586 channel
#      area = 59.473 K channel
f.store_fit('orions_hc3n_fit.txt')      # store fit
# Save the spectrum
spave.save('orions_hc3n_reduced','ASCII',True) # save the spectrum

```

A.4 Single Dish Imaging

Single dish imaging is supported within CASA using standard tasks and tools. The data must be in the Measurement Set format. Once there, you can use the `im` (imager) tool to create images:

Tool example:

```

scans.save('outputms','MS2')           # Save your data from ASAP into an MS

im.open('outputms')                    # open the data set
im.selectvis(nchan=901,start=30,step=1, # choose a subset of the data
            spwid=0,field=0)           # (just the key emission channels)
dir='J2000 17:18:29 +59.31.23'        # set map center
im.defineimage(nx=150,cellx='1.5arcmin', # define image parameters
              phasecenter=dir,mode='channel',start=30, # (note it assumes symmetry if ny,celly
              nchan=901,step=1)        # aren't specified)

im.setoptions(ftmachine='sd',cache=100000000) # choose SD gridding
im.setsdoptions(convsupport=4)          # use this many pixels to support the
                                        # gridding function used
                                        # (default=prolate spheroidal wave function)

im.makeimage(type='singledish',        # make the image
             image='FLS3a_HI.image')

```

A.4.1 Single Dish Imaging Use Case With ASAP Toolkit

The data summary and the script are given below.

```

# Project: AGBT02A_007_01
# Observation: GBT(1 antennas)
#
# Telescope Observation Date   Observer   Project

```

```

#   GBT      [                4.57539e+09, 4.5754e+09]Lockman      AGBT02A_007_01
#   GBT      [                4.57574e+09, 4.57575e+09]Lockman      AGBT02A_007_02
#   GBT      [                4.5831e+09, 4.58313e+09]Lockman      AGBT02A_031_12
#
# Thu Feb 1 23:15:15 2007    NORMAL ms::summary:
# Data records: 76860        Total integration time = 7.74277e+06 seconds
#   Observed from 22:05:41   to   12:51:56
#
# Thu Feb 1 23:15:15 2007    NORMAL ms::summary:
# Fields: 2
#   ID   Name           Right Ascension  Declination   Epoch
#   0    FLS3a          17:18:00.00    +59.30.00.00 J2000
#   1    FLS3b          17:18:00.00    +59.30.00.00 J2000
#
# Thu Feb 1 23:15:15 2007    NORMAL ms::summary:
# Spectral Windows: (2 unique spectral windows and 1 unique polarization setups)
#   SpwID  #Chans Frame Ch1(MHz)   Resoln(kHz) TotBW(kHz)  Ref(MHz)   Corrs
#   0      1024 LSRK  1421.89269  2.44140625  2500        1420.64269 XX  YY
#   1      1024 LSRK  1419.39269  2.44140625  2500        1418.14269 XX  YY

# FLS3 data calibration
# this is calibration part of FLS3 data
#
casapath=os.environ['AIPSPATH']
import asap as sd
os.environ['AIPSPATH']=casapath

print '--Import--'

s=sd.scantable('FLS3_all_newcal_SP',false)      # read in MeasurementSet

print '--Split--'

# splitting the data for each field
s0=s.get_scan('FLS3a*')                        # split the data for the field of interest
s0.save('FLS3a_HI.asap')                       # save this scantable to disk (asap format)
del s0                                          # free up memory from scantable

print '--Calibrate--'
s=sd.scantable('FLS3a_HI.asap')               # read in scantable from disk (FLS3a)
s.set_fluxunit('K')                           # set the brightness units to Kelvin
scanns = s.getscannos()                       # get a list of scan numbers
sn=list(scanns)                               # convert it to a list
print "No. scans to be processed:", len(scanns)

res=sd.calfs(s,sn)                            # calibrate all scans listed using frequency
                                           # switched calibration method

print '--Save calibrated data--'
res.save('FLS3a_calfs', 'MS2')                # Save the dataset as a MeasurementSet

```

```

print '--Image data--'

im.open('FLS3a_calfs') # open the data set
im.selectvis(nchan=901,start=30,step=1, # choose a subset of the data
            spwid=0,field=0) # (just the key emission channels)
dir='J2000 17:18:29 +59.31.23' # set map center
im.defineimage(nx=150,cellx='1.5arcmin', # define image parameters
              phasecenter=dir,mode='channel',start=30, # (note it assumes symmetry if ny,celly
              nchan=901,step=1) # aren't specified)

im.setoptions(ftmachine='sd',cache=100000000) # choose SD gridding
im.setsoptions(convsupport=4) # use this many pixels to support the
                                # gridding function used
                                # (default=prolate spheroidal wave function)
im.makeimage(type='singledish',image='FLS3a_HI.image') # make the image

```

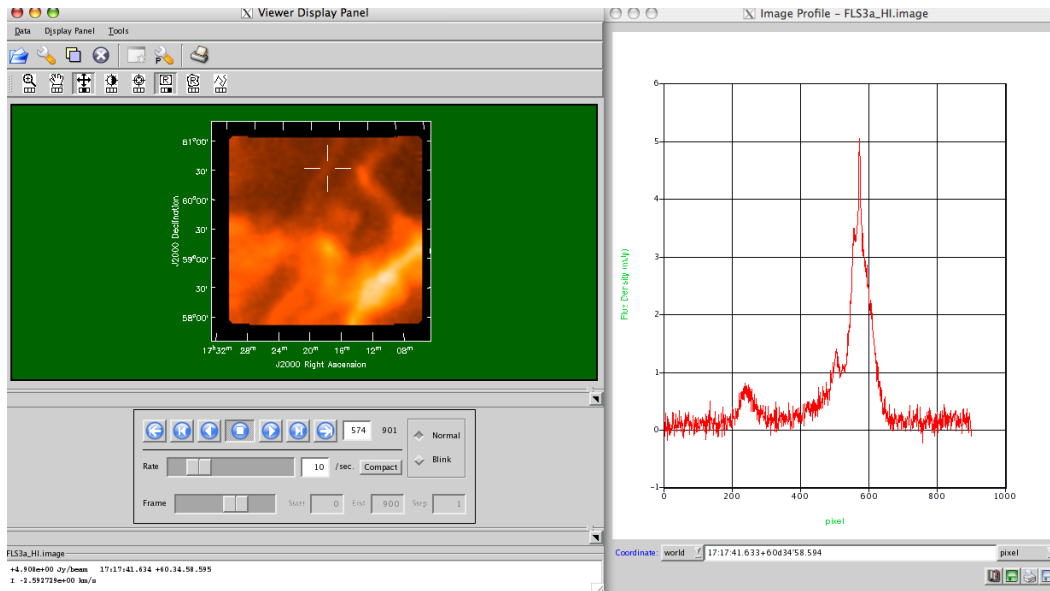


Figure A.6: FLS3a HI emission. The display illustrates the visualization of the data cube (left) and the profile display of the cube at the cursor location (right); the Tools menu of the Viewer Display Panel has a Spectral Profile button which brings up this display. By default, it grabs the left-mouse button. Pressing down the button and moving in the display will show the profile variations.

A.5 Known Issues, Problems, Deficiencies and Features

The Single-Dish calibration and analysis package within CASA is still very much under development. Not surprisingly, there are a number of issues with ASAP and the SDtasks that are known and under repair. Some of these are non-obvious "features" by the way ASAP and `sd` are implemented, or limitations of the current Python tasking environment. Some are functions that have yet to be implemented. These currently include:

1. `sd.plotter`

Currently one can get hardcopy only after making a viewed plot. Ideally, ASAP should allow choosing the plotting device when you set up the plotter.

Multi-panel plotting is poor. Currently one can only add things (like lines, text, etc.) to the first panel. Also, `sd.plotter.set_range()` sets the same range for multiple panels, while we would like it to be able to set the range for each independently, including the default ranges.

The appearance of the plots need to be improved. In principle, matplotlib can make "publication quality" figures, but in practice a lot of work is required and the plots are not good.

The `sd.plotter` object remembers things throughout the session and thus can easily get confused. For example, one must reset the range `sd.plotter.set_range()` if set manually. This behavior is not always expected, but is a consequence of having `sd.plotter` be its own object that is fed data and commands.

There are known bugs when switching between the task-based plotting with `sdplot` and the tool-based plotting. `sdplot` adds extra control buttons that are not implemented in the `sd` toolkit, so there are no 'spec value', 'statistics', or 'Quit' buttons for `sd.plotter.plot()`. Once the 'statistics' button on `sdplot` is used and deleted, `sdplot`, you can still use mouse buttons to select region to get statistics with the plotter displayed with `sd.plotter.plot()`. But, spectral values won't be displayed in such case. Also, when going back to `sdplot` after using `sd.plotter.plot()`, an error may occur and the plotter window won't be displayed. In that case, the window state can be reset by deleting the plotter window opened by `sd.plotter.plot()` with the command, `sd.plotter._plotter.unmap()`.

Eventually we would like the capability to interactively set things using the plots, like select frequency ranges, identify lines, start fitting.

2. `sd.selector`

The selector object only allows one selection of each type. It would be nice to be able to make a union of selections (without resorting to query) for the `set_name`. Note that the others like scans and IFs work off lists, which is fine. We should make `set_name` work off lists of names.

3. `sd.scantable`

There is no useful inline help on the scantable constructor in `help sd.scantable help sd`.

The inline help for `scantable.summary` claims that there is a `verbose` parameter, but there is not. The `scantable.verbosesummary` `asaprc` parameter (e.g. in `sd.rcParams`) does nothing.

GBT data has an undefined parameter fluxunit ('' that should be 'K'), an incorrect freqframe ('LSRK' that is really 'TOPO'), and reference frequency (set to that of the first IF only).

The `sd.scantable.freq_align` does not yet work correctly.

4. `sd` general issues

There should be a `sdhelp` equivalent of `toolhelp` and `tasklist` for the `sd` tools and tasks.

The current output of ASAP is verbose and controlled by setting `sd.rcParams['verbose']=False` (or `True`). We will make some of the output less cryptic.

We will strip off leading and trailing whitespace on string parameters.

5. `SDtasks` general issues

The `SDtasks` work with files saved onto disk in one of the `scantable` supported formats. It might be useful to work with `scantables` in memory (passing the objects) but this would require changes to the tasking system. Note that this behavior is consistent throughout the `casapy` tasks.

6. `sdaverage` (and `sdcal`)

`averageall=True` is still experimental since the test was insufficient because of a lack of test data.

7. `sdfit`

Only way to handle multi-IFs is to set `fitmode='auto'` (linefinder is applied for each spectra and derives initial guesses). For `fitmode='list'`, there are no way to give initial guesses for each IFs by hand.

8. `sdplot`

Only handles the included JPL line catalog. Also, see `sd.plotter` issues above.

Appendix B

Simulation

New in 3.0.1: The angular size of the simulated map must be defined by the parameter *mosaicsize*. See §B.1.

The capability for simulating observations and datasets from the EVLA and ALMA are an important use-case for CASA. This not only allows one to get an idea of the capabilities of these instruments for doing science, but also provides benchmarks for the performance and utility of the software for processing “realistic” datasets (with atmospheric and instrumental effects). CASA can now calculate visibilities (create a measurement set) for any interferometric array, and calculate and apply calibration tables representing some of the most important corrupting effects. One can use the `sdsim` task to simulate single dish or total power data. (the ASAP package must be loaded, see Sect. A).

CASA’s simulation capabilities have been under development, and will continue to evolve through this and the next release of CASA. For the most current information, please refer to <http://www.casaguides.nrao.edu>, and click on “Simulating Observations in CASA”. Following general CASA practice, the greatest flexibility and richest functionality is at the Toolkit level. The most commonly used procedures for interferometric simulation are encapsulated in the `simdata` task. Total power simulation is possible at the task level using the `sdsim` task, which follows somewhat parallel inputs and whose capabilities will be developed more fully in the next release.

Inside the Toolkit:

The simulator methods are in the `sm` tool. Many of the other tools are also helpful when constructing and analyzing simulations.

B.1 Simulating ALMA with `simdata`

The inputs are:

```
# simdata :: mosaic simulation task:
project          =      'sim'          # root for output files
```

```

complist          =          ''          # [optional] componentlist table to observe
modelimage       =          ''          # model sky image name
inbright         = 'unchanged'         # set peak surface brightness in Jy/pixel or "unchanged"
ignorecoord      =          False       # change model coordinates
startfreq        = '89GHz'             # [only if ignorecoord=T] frequency of first channel
chanwidth        = '10MHz'            # [only if ignorecoord=T] channel width
refdate          = '2012/05/21/22:05:00' # center time/date of observation *see help
totaltime        = '7200s'            # total time of observation
integration       = '10s'             # integration (sampling) time
scanlength       =          5          # number of integrations per pointing in the mosaic
direction        = ['J2000 19h00m00 -40d00m00'] # mosaic center, or list of pointings
pointingspacing  = '1arcmin'          # spacing in between beams in mosaic
mosaicsize       = ['1.0arcmin', '1.0arcmin'] # angular size of desired area to map [*NEW*]
caldirection     =          ''          # pt source calibrator [experimental]
calflux          = '1Jy'              #
checkinputs      = 'no'               # graphically verify parameters [yes|no|only]
antennalist      = 'alma.out10.cfg'    # antenna position file
noise_thermal    =          True        # add thermal noise
    noise_mode    = 'tsys-atm'         # tsys-atm: set PWV and use ATM library; tsys-manual:
    # set t_sky and tau
    user_pwv      =          1.0        # Precipitable Water Vapor in mm [tsys-atm only]
    t_ground      =          269.0      # ambient temperature
    t_sky         =          263.0      # atmospheric temperature [tsys-manual only]
    tau0          =          0.1        # zenith opacity [tsys-manual only]
cell             = '0.1arcsec'         # output cell/pixel size
imsize          = [128, 128]          # output image size in pixels (x,y)
threshold        = '0.01mJy'         # flux level (+units) to stop cleaning
niter           =          500         # maximum number of iterations
psfmode         = 'clark'             # minor cycle PSF calculation method
weighting       = 'natural'          # weighting to apply to visibilities
uvtaper         =          False       # apply additional uv tapering of visibilities.
stokes          = 'I'                # Stokes params to image
fidelity         =          True        # Calculate fidelity images
display         =          True        # Plot simulation result images,figures
verbose         =          False

```

This task takes an input model image or list of components, plus a list of antennas (locations and sizes), and simulates a particular observation (specifies by mosaic setup and observing cycles and times). The output is a MS suitable for further analysis in CASA, a synthesized image create from those visibilities, a difference image between the synthesized image and your sky model convolved with the output synthesized beam, and a fidelity image.

Random thermal noise (from the atmosphere and receiver) can optionally be added. A realistic model of the troposphere is created from known site characteristics (altitude, etc) and used to calculate the frequency dependence of the noise. Similarly, specifications for ALMA and EVLA receiver temperature and antenna efficiencies are coded into the task to determine a realistic system temperature. Other corrupting effects, such as phase noise, gain drift, and cross-polarization, can be added the the MS with the toolkit - see the web documentation for more information.

Much more detailed information, and use-case examples, can be found at

http://casaguides.nrao.edu/index.php?title=Simulating_Observations_in_CASA

Appendix C

Obtaining and Installing CASA

C.1 Installation Script

Currently you must be able to log into your system as the root user or an administrator user to install CASA.

The easiest way to install CASA on a RedHat Enterprise Linux (or compatible) system is to use our installation script, `load-casapy`. This script will ftp the CASA RPMs and install them. To use it, first use the link above to download it to your hard disk. Next, make sure execute permission is set for the file.

Install CASA into `/usr` by logging in as root and running:

```
load-casapy -root
```

This option will install CASA into `/usr`, but it can only be run by the root user.

Alternatively, you can visit our FTP server, download the rpms, and install them by hand. Note: you must be root/administrator to install CASA in this manner.

See the following for more details:

<https://wikio.nrao.edu/bin/view/Software/ObtainingCASA>

C.2 Startup

This section assumes that CASA has been installed on your LINUX or OSX system. *For NRAO-AOC testers, you should do the following on an AOC RHE4 machine:*

```
> . /home/casa/casainit.sh  
or  
> source /home/casa/casainit.csh
```

Appendix D

Python and CASA

CASA uses Python, IPython and matplotlib within the package. IPython is an enhanced, interactive shell to Python which provides many features for efficient command line interaction, while matplotlib is a Python 2-D plotting library for publication quality figures in different hardcopy formats.

From www.python.org: "Python is an interpreted, interactive, object-oriented programming language". Python is used as the underlying command line interface/scripting language to CASA. Thus, CASA inherits the features and the annoyances of Python. For example, since Python is inherently 0-based in its indexing of arrays, vectors, etc, CASA is also 0-based; any Index inputs (e.g., start (for start channel), fieldIndex, antennaID, etc) will start with 0. Another example is that indenting of lines means something to Python, of which users will have to be aware.

Some key links are:

- <http://python.org> – Main Python page
- <http://python.org/doc/2.4.2/ref/ref.html> – Python Reference
- <http://python.org/doc/2.4.2/tut/tut.html> – Python Tutorial
- <http://ipython.scipy.org> – IPython page
- <http://matplotlib.sourceforge.net> – matplotlib page

Each of the features of these components behave in the standard way within CASA . In the following sections, we outline the key elements for analysis interactions; see the Python references and the IPython page for the full suite of functionality.

D.1 Automatic parentheses

Automatic parenthesis is enabled for calling functions with argument lists; this feature is intended to allow less typing for common situations. IPython will display the interpretation of the line,

beneath the one typed, as indicated by the '----->'. Default behavior in CASA is to have automatic parenthesis enabled.

D.2 Indentation

Python pays attention to indentation of lines in scripts or when you enter them interactively. It uses indentation to determine the level of nesting in loops. Be careful when cutting and pasting, if you get the wrong indentation, then unpredictable things can happen (usually it just gives an error).

A blank line can be used to return the indentation to a previous level. For example, expanded parameters in tasks cause indentation in subsequent lines in the interface. For example, the following snippet of inputs from `clean` can be cut and pasted without error due to the blank line after the indented parameters:

```
mode          = 'channel'      # Type of selection
  nchan       =      -1        # Number of channels to select
  start       =         0      # Start channel
  step        =         1      # Increment between channels/velocity
  width       =         1      # Channel width

alg           = 'clark'       # Algorithm to use
```

If the blank line were not there, an error would result if you pasted this at the `casapy` prompt.

D.3 Lists and Ranges

Sometimes, you need to give a task a list of indices. For example, some tasks and tools expect a comma-separated Python list, e.g.

```
scanlist = [241, 242, 243, 244, 245, 246]
```

You can use the Python `range` function to generate a list of consecutive numbers, e.g.

```
scanlist = range(241,247)
```

giving the same list as above, e.g.

```
CASA <1>: scanlist=range(241,247)
CASA <2>: print scanlist
[241, 242, 243, 244, 245, 246]
```

Note that `range` starts from the first limit and goes to one below the second limit (Python is 0-based, and `range` is designed to work in loop functions). If only a single limit is given, the first limit is treated as 0, and the one given is used as the second, e.g.

```
CASA <3>: iflist=range(4)
CASA <4>: print iflist
[0, 1, 2, 3]
```

You can also combine multiple ranges by summing lists

```
CASA <5>: scanlist=range(241,247) + range(251,255)
CASA <6>: print scanlist
[241, 242, 243, 244, 245, 246, 251, 252, 253, 254]
```

D.4 Dictionaries

Python dictionaries are data structures that contain `key:value` pairs, sort of like a hash array. These are useful to store mini-databases of things. In CASA, the parameter values are kept in a dictionary behind the scenes.

To initialize a dictionary, say we call it `mydict`, for use:

```
CASA <7>: mydict = {}
```

To add members:

```
CASA <8>: mydict['source'] = '0137+331'
CASA <9>: mydict['flux'] = 5.4
```

To see its contents:

```
CASA <10>: mydict
Out[10]: {'flux': 5.4000000000000004, 'source': '0137+331'}
CASA <11>: print mydict
{'source': '0137+331', 'flux': 5.4000000000000004}
```

To access a specific entry:

```
CASA <12>: print mydict['flux']
5.4
```

D.4.1 Saving and Reading Dictionaries

To save a simple dictionary to a file:

```
CASA <13>: dictfile = open('mydictfile.py','w')
CASA <14>: print >>dictfile,"mydict = ",mydict
CASA <15>: dictfile.close()
CASA <16>: !cat mydictfile.py
```



```
IPython system call: cat mydictfile.py
mydict = {'source': '0137+331', 'flux': 5.4000000000000004}
```

```
CASA <17>: mydict = {}
CASA <18>: run mydictfile.py
CASA <19>: mydict
  Out[19]: {'flux': 5.4000000000000004, 'source': '0137+331'}
```

More complex dictionaries, like those produced by `imstat` that contain NumPy arrays, require a different approach to save. The `pickle` module lets you save general data structures from Python. For example:

```
CASA <20>: import pickle
CASA <21>: xstat
  Out[21]:
{'blc': array([0, 0, 0, 0]),
 'blcf': '15:24:08.404, +04.31.59.181, I, 1.41281e+09Hz',
 'flux': array([ 4.0795296]),
 'max': array([ 0.05235516]),
 'maxpos': array([134, 134,  0, 38]),
 'maxposf': '15:21:53.976, +05.05.29.998, I, 1.41374e+09Hz',
 'mean': array([ 1.60097857e-05]),
 'medabsdevmed': array([ 0.00127436]),
 'median': array([ -1.17422514e-05]),
 'min': array([-0.0104834]),
 'minpos': array([160,  1,  0, 30]),
 'minposf': '15:21:27.899, +04.32.14.923, I, 1.41354e+09Hz',
 'npts': array([ 3014656.]),
 'quartile': array([ 0.00254881]),
 'rms': array([ 0.00202226]),
 'sigma': array([ 0.0020222]),
 'sum': array([ 48.26399646]),
 'sumsq': array([ 12.32857318]),
 'trc': array([255, 255,  0, 45]),
 'trcf': '15:19:52.390, +05.35.44.246, I, 1.41391e+09Hz'}
```

```
CASA <22>: mydict
  Out[22]: {'flux': 5.4000000000000004, 'source': '0137+331'}
```

```
CASA <23>: pickfile = 'myxstat.pickle'
CASA <24>: f = open(pickfile, 'w')
CASA <25>: p = pickle.Pickler(f)
CASA <26>: p.dump(xstat)
CASA <27>: p.dump(mydict)
CASA <28>: f.close()
```

The dictionaries are now saved in pickle file `myxstat.pickle` in the current directory.

To retrieve:

```
CASA <29>: xstat2 = {}
```

```

CASA <30>: mydict2 = {}
CASA <31>: f = open(pickfile)
CASA <32>: u = pickle.Unpickler(f)
CASA <33>: xstat2 = u.load()
CASA <34>: mydict2 = u.load()
CASA <35>: f.close()
CASA <36>: xstat2
  Out[36]:
{'blc': array([0, 0, 0, 0]),
 'blcf': '15:24:08.404, +04.31.59.181, I, 1.41281e+09Hz',
 'flux': array([ 4.0795296]),
 'max': array([ 0.05235516]),
 'maxpos': array([134, 134,  0,  38]),
 'maxposf': '15:21:53.976, +05.05.29.998, I, 1.41374e+09Hz',
 'mean': array([ 1.60097857e-05]),
 'medabsdevmed': array([ 0.00127436]),
 'median': array([ -1.17422514e-05]),
 'min': array([-0.0104834]),
 'minpos': array([160,  1,  0,  30]),
 'minposf': '15:21:27.899, +04.32.14.923, I, 1.41354e+09Hz',
 'npts': array([ 3014656.]),
 'quartile': array([ 0.00254881]),
 'rms': array([ 0.00202226]),
 'sigma': array([ 0.0020222]),
 'sum': array([ 48.26399646]),
 'sumsq': array([ 12.32857318]),
 'trc': array([255, 255,  0,  45]),
 'trcf': '15:19:52.390, +05.35.44.246, I, 1.41391e+09Hz'}

CASA <37>: mydict2
  Out[37]: {'flux': 5.4000000000000004, 'source': '0137+331'}

```

Thus, you can make scripts that save information and use it later, like for regressions.

Note that these examples use Python file-handling and IO, as well as importing modules such as `pickle`. See your friendly Python reference for more on this kind of stuff. Its fairly obvious how it works.

D.5 Control Flow: Conditionals, Loops, and Exceptions

There are a number of ways to control the flow of execution in Python, including conditionals (`if`), loops (`for` and `while`), and exceptions (`try`). We will discuss the first two below.

D.5.1 Conditionals

The standard `if` block handles conditional execution or branches in Python:

```

if <expression>:
    <statements>
elif <expression>:
    <statements>
elif <expression>:
    <statements>
...
else:
    <statements>

```

Insert a `pass` statement if you want no action to be taken for a particular clause. The `<expression>` should reduce down to `True` or `False`.

For example,

```

if ( importmode == 'vla' ):
    # Import the data from VLA Export to MS
    default('importvla')
    print "Use importvla to read VLA Export and make an MS"

    archivefiles = datafile
    vis = msfile
    bandname = exportband
    autocorr = False
    antnamescheme = 'new'
    project = exportproject

    importvla()
elif ( importmode == 'fits' ):
    # Import the data from VLA Export to MS
    default('importuvfits')
    print "Use importuvfits to read UVFITS and make an MS"

    fitsfile = datafile
    vis = msfile
    async = False

    importuvfits()
else:
    # Copy from msfile
    print "Copying "+datafile+" to "+msfile
    os.system('cp -r '+datafile+' '+msfile)
    vis = msfile

```

chooses branches based on the value of the `importmode` Python variable (set previously in script).

D.5.2 Loops

The for loop

```
for iter in seq:
    <statements>
```

iterates over elements of a sequence `seq`, assigning each in turn to `iter`. The sequence is usually a list of values.

For example,

```
splitms = 'polcal_20080224.cband.all.split.ms'
srclist = ['0137+331', '2136+006', '2202+422', '2253+161', '0319+415', '0359+509']
spwlist = ['0', '1']
```

```
for src in srclist:

    for spwid in spwlist:

        imname = splitms + '.' + src + '.' + spwid + '.clean'
        clean(vis=splitms, field=src, spw=spwid, imagename=imname,
              stokes='IQUV', psfmode='hogbom', imagermode='csclean',
              imsize=[288,288], cell=[0.4,0.4], niter=1000,
              threshold=1.3, mask=[134,134,154,154])
```

```
    # Done with spw
```

```
# Done with sources
```

As usual, blocks are closed by blank lines of the previous indentation level.

You can use the `range` (§ D.3) Python function to generate a numerical loop:

```
vis = 'polcal_20080224.cband.all.ms'
for i in range(0,6):
    fld = str(i)
    plotxy(vis, field=fld, xaxis='uvdist', yaxis='amp')
```

```
# Done with fields [0, 1, 2, 3, 4, 5]
```

There is also a `while` loop construct

```
while <expression>:
    <statements>
```

which executes the statement block while the `<expression>` is `True`. The `while` loop can also take an `else` block.

For example,

```
# Do an explicit set of clean iterations down to a limit
prevrms = 1.e10
while rms > 0.001 :
    clean(vis=splitms,field=src,spw=spwid,imagename=imname,
          stokes='IQUV',psfmode='hogbom',imagermode='csclean',
          imsize=[288,288],cell=[0.4,0.4],niter=200,
          threshold=1.3,mask=[134,134,154,154])

    offstat=imstat(imname+'.residual',box='224,224,284,284')
    rms=offstat['sigma'][0]
    if rms > prevrms:
        break                # the rms has increased, stop

    prevrms = rms

# Clean until the off-source rms residual, reaches 0.001 Jy
```

Note that you can exit a loop using the `break` statement, as we have here when the rms increases.

D.6 System shell access

For scripts, the `os.system` methods are the preferred way to access system shell commands (see § D.6.1).

In interactive mode, any input line beginning with a `!` character is passed verbatim (minus the `!`) to the underlying operating system. Several common commands (`ls`, `pwd`, `less`) may be executed with or without the `!`. Note that the `cd` command must be executed without the `!`, and the `cp` command must use `!!` as there is a conflict with the `cp` tool in `casapy`.

For example:

```
CASA [1]: pwd
/export/home/corsair-vml/jmcmulli/data
CASA [2]: ls n*
ngc5921.ms ngc5921.py
CASA [3]: !cp -r ../test.py .
```

D.6.1 Using the `os.system` methods

To use this, you need the `os` package. This should be loaded by default by `casapy`, but if not you can use

```
import os
```

in your script.

For example, in our scripts we use this to clean up any existing output files

```
# The prefix to use for all output files
prefix='ngc5921.usecase'
```

```
# Clean up old files
os.system('rm -rf '+prefix+'*')
```

Note that the `os` package has many useful methods. You can see these by using tab-completion:

```
CASA <2>: os.<tab>
Display all 223 possibilities? (y or n)
os.EX_CANTCREAT      os.X_OK              os.fdatasync         os.readlink
os.EX_CONFIG        os._Environ          os.fdopen            os.remove
os.EX_DATAERR       os.__all__           os.fork              os.removedirs
os.EX_IOERR         os.__class__         os.forkpty           os.rename
os.EX_NOHOST        os.__delattr__       os.fpathconf         os.renames
os.EX_NOINPUT       os.__dict__          os.fstat             os.rmdir
os.EX_NOPERM        os.__doc__           os.fstatvfs          os.sep
os.EX_NOUSER        os.__file__          os.fsync             os.setegid
os.EX_OK            os.__getattr__       os.ftruncate         os.seteuid
os.EX_OSERR         os.__hash__          os.getcwd            os.setgid
os.EX_OSFILE        os.__init__          os.getcwdu           os.setgroups
os.EX_PROTOCOL      os.__name__          os.getegid           os.setpgid
os.EX_SOFTWARE      os.__new__           os.getenv            os.setpgrp
os.EX_TEMPFAIL      os.__reduce__        os.geteuid           os.setregid
os.EX_UNAVAILABLE   os.__reduce_ex__    os.getgid            os.setreuid
os.EX_USAGE         os.__repr__          os.getgroups         os.setsid
os.F_OK             os.__setattr__       os.getloadavg        os.setuid
os.NGROUPS_MAX      os.__str__           os.getlogin          os.spawnl
os.O_APPEND         os._copy_reg         os.getpgid           os.spawnle
os.O_CREAT          os._execvpe          os.getpgrp           os.spawnlp
os.O_DIRECT         os._exists           os.getpid            os.spawnlpe
os.O_DIRECTORY      os._exit             os.getppid           os.spawnv
os.O_DSYNC          os._get_exports_list os.getsid            os.spawnve
os.O_EXCL           os._make_stat_result os.getuid            os.spawnvp
os.O_LARGEFILE      os._make_statvfs_result os.isatty           os.spawnvpe
os.O_NDELAY         os._pickle_stat_result os.kill              os.stat
os.O_NOCTTY         os._pickle_statvfs_result os.killpg           os.stat_float_times
os.O_NOFOLLOW       os._spawnvef         os.lchown            os.stat_result
os.O_NONBLOCK       os.abort             os.linesep           os.statvfs
os.O_RDONLY         os.access            os.link              os.statvfs_result
os.O_RDWR           os.altsep            os.listdir           os.strerror
os.O_RSYNC          os.chdir             os.lseek             os.symlink
os.O_SYNC           os.chmod             os.lstat             os.sys
os.O_TRUNC          os.chown             os.major             os.sysconf
os.O_WRONLY         os.chroot            os.makedev           os.sysconf_names
os.P_NOWAIT         os.close             os.makedirs          os.system
os.P_NOWAITO        os.confstr           os.minor             os.tcgetpgrp
os.P_WAIT           os.confstr_names     os.mkdir             os.tcsetpgrp
os.R_OK             os.ctermid           os.mkfifo            os.tmpnam
os.SEEK_CUR         os.curdir            os.mknod             os.times
os.SEEK_END         os.defpath           os.name              os.tmpfile
```

os.SEEK_SET	os.devnull	os.nice	os.tmpnam
os.TMP_MAX	os.dup	os.open	os.ttyname
os.UserDict	os.dup2	os.openpty	os.umask
os.WCONTINUED	os.environ	os.pardir	os.uname
os.WCOREDUMP	os.error	os.path	os.unlink
os.WEXITSTATUS	os.execl	os.pathconf	os.unsetenv
os.WIFCONTINUED	os.execl	os.pathconf_names	os.urandom
os.WIFEXITED	os.execlp	os.pathsep	os.utime
os.WIFSIGNALED	os.execlpe	os.pipe	os.wait
os.WIFSTOPPED	os.execv	os.popen	os.wait3
os.WNOHANG	os.execve	os.popen2	os.wait4
os.WSTOPSIG	os.execvp	os.popen3	os.waitpid
os.WTERMSIG	os.execvpe	os.popen4	os.walk
os.WUNTRACED	os.extsep	os.putenv	os.write
os.W_OK	os.fchdir	os.read	

D.6.2 Directory Navigation

In addition, filesystem navigation is aided through the use of bookmarks to simplify access to frequently-used directories:

```
CASA [4]: cd /home/ballista/jmcmulli/other_data
CASA [4]: pwd
/home/ballista/jmcmulli/other_data
CASA [5]: bookmark other_data
CASA [6]: cd /export/home/corsair-vml/jmcmulli/data
CASA [7]: pwd
/export/home/corsair-vml/jmcmulli/data
CASA [8]: cd -b other_data
(bookmark:data) -> /home/ballista/jmcmulli/other_data
```

D.6.3 Shell Command and Capture

See also § D.8 for the use of the command history.

1. `sx shell_command`, `!!shell_command` - this captures the output to a list

```
CASA [1]: sx pwd # stores output of 'pwd' in a list
Out[1]: ['/home/basho3/jmcmulli/pretest']

CASA [2]: !!pwd # !! is a shortcut for 'sx'
Out[2]: ['/home/basho3/jmcmulli/pretest']

CASA [3]: sx ls v* # stores output of 'pwd' in a list
Out[3]:
['vla_calplot.jpg',
'vla_calplot.png',
'vla_msplot_cals.jpg',
```

```
'vla_msplot_cals.png',
'vla_plotcal_bpass.jpg',
'vla_plotcal_bpass.png',
'vla_plotcal_fcal.jpg',
'vla_plotcal_fcal.png',
'vla_plotvis.jpg',
'vla_plotvis.png']
```

CASA [4]: x=_ # remember '_' is a shortcut for the output from the last command

```
CASA [5]: x
Out[5]:
['vla_calplot.jpg',
'vla_calplot.png',
'vla_msplot_cals.jpg',
'vla_msplot_cals.png',
'vla_plotcal_bpass.jpg',
'vla_plotcal_bpass.png', 'vla_plotcal_fcal.jpg',
'vla_plotcal_fcal.png',
'vla_plotvis.jpg',
'vla_plotvis.png']
```

CASA [6]: y=Out[2] # or just refer to the enumerated output

```
CASA [7]: y
Out[7]: ['/home/basho3/jmcmulli/pretest']
```

2. sc - captures the output to a variable; options are '-l' and '-v'

CASA [1]: sc x=pwd # capture output from 'pwd' to the variable 'x'

```
CASA [2]: x
Out[2]: '/home/basho3/jmcmulli/pretest'
```

CASA [3]: sc -l x=pwd # capture the output from 'pwd' to the variable 'x' but
split newlines into a list (similar to sx command)

```
CASA [4]: x
Out[4]: ['/home/basho3/jmcmulli/pretest']
```

CASA [5]: sc -v x=pwd # capture output from 'pwd' to a variable 'x' and
show what you get (verbose mode)

```
x ==
'/home/basho3/jmcmulli/pretest'
```

```
CASA [6]: x
Out[6]: '/home/basho3/jmcmulli/pretest'
```


D.7 Logging

There are two components to logging within CASA. Logging of all command line inputs is done via IPython.

Upon startup, CASA will log all commands to a file called `ipython.log`. This file can be changed via the use of the `ipythonrc` file. This log file can be edited and re-executed as appropriate using the `execfile` feature (§ D.11).

The following line sets up the logging for CASA . There are four options following the specification of the logging file: 1) `append`, 2) `rotate` (each session of CASA will create a new log file with a counter incrementing `ipython.log.1`, `ipython.log.2` etc, 3) `over` (overwrite existing file), and 4) `backup` (renames existing log file to `log_name`).

```
logfile ./ipython.log append
```

The command `logstate` will provide details on the current logging setup:

```
CASA [12]: logstate

File:   ipython.log
Mode:   append
State:  active
```

Logging can be turned on and off using the `logon`, `logoff` commands.

The second component is the output from applications which is directed to the file `./casapy.log`. See § 1.4.2 for more on the `casalogger`.

D.8 History and Searching

Numbered input/output history is provided natively within IPython. Command history is also maintained on-line.

```
CASA [11]: x=1

CASA [12]: y=3*x

CASA [13]: z=x**2+y**2

CASA [14]: x
Out[14]: 1

CASA [15]: y
Out[15]: 3
```

```

CASA [16]: z
Out[16]: 10

CASA [17]: Out[14] # Note: The 'Out' vector contains command output
Out[17]: 1

CASA [18]: _15 # Note: The return value can be accessed by _number
Out[18]: 3

CASA [19]: ___ # Note: The last three return values can be accessed as:
Out[19]: 10 # --, ---, ---

```

Command history can be accessed via the 'hist' command. The history is reset at the beginning of every CASA session, that is, typing 'hist' when you first start CASA will not provide any commands from the previous session. However, all of the commands are still available at the command line and can be accessed through the up or down arrow keys, and through searching.

```

CASA [22]: hist
1 : __IP.system("vi temp.py") # Note:shell commands are designated in this way
2 : ipmagic("run -i temp.py") # Note:magic commands are designated in this way
3 : ipmagic("hist ")
4 : more temp.py
5 : __IP.system("more temp.py")
6 : quickhelp() # Note: autoparenthesis are added in the history
7 : im.open('ngc5921.ms')
8 : im.summary()
9 : ipmagic("pdoc im.setdata")
10: im.close()
11: quickhelp()
12: ipmagic("logstate ")
13: x=1
14: y=3*x
15: z=x**2+y**2
16: x
17: y
18: z
19: Out[16]
20: _17
21: ___

```

The history can be saved as a script or used as a macro for further use:

```

CASA [24]: save script.py 13:16
File 'script.py' exists. Overwrite (y/[N])? y
The following commands were written to file 'script.py':
x=1
y=3*x
z=x**2+y**2

```

```
CASA [25]: !more script.py
x=1
y=3*x
z=x**2+y**2
```

Note that the history commands will be saved up to, but not including the last value (i.e., history commands 13-16 saves commands 13, 14, and 15).

There are two mechanisms for searching command history:

1. Previous/Next: use `Ctrl-p` (previous,up) and `Ctrl-n` (next,down) to search through only the history items that match what you have typed so far (min-match completion). If you use `Ctrl-p` or `Ctrl-n` at a blank prompt, they behave just like the normal arrow keys.
2. Search: `Ctrl-r` opens a search prompt. Begin typing and the system searches your history for lines that contain what you've typed so far, completing what it can. For example:

```
CASA [37]: <Ctrl-r>

(reverse-i-search) '':
```

Typing anything after the colon will provide you with the last command matching the characters, for example, typing 'op' finds:

```
(reverse-i-search) 'op': im.open('ngc5921.ms')
```

Subsequent hitting of `Ctrl-r` will search for the next command matching the characters.

D.9 Macros

Macros can be made for easy re-execution of previous commands. For example to store the commands 13-15 to the macro 'example':

```
CASA [31]: macro example 13:16
Macro 'example' created. To execute, type its name (without quotes).
Macro contents:
x=1
y=3*x
z=x**2+y**2
```

```
CASA [32]: z
Out[32]: 6
```

```
CASA [33]: z=10
```

```
CASA [34]: example
Out[34]: Executing Macro...
```

```
CASA [35]: z
Out[35]: 6
```

```
CASA [36]:
```

D.10 On-line editing

You can edit files on-line in two ways:

1. Using the shell access via '!vi'
2. Using the ed function; this will edit the file but upon closing, it will try to execute the file; using the 'script.py' example above:

```
CASA [13]: ed script.py # this will bring up the file in your chosen editor
                # when you are finished editing the file,
                # it will automatically
                # execute it (as though you had done a
                # execfile 'script.py'
```

```
Editing... done. Executing edited code...
```

```
CASA [14]: x
Out[14]: 1
```

```
CASA [15]: y
Out[15]: 3
```

```
CASA [16]: z
Out[16]: 6
```

D.11 Executing Python scripts

Python scripts are simple text files containing lists of commands as if typed at the keyboard. Note: the auto-parentheses feature of IPython can not be used in scripts, that is, you should make sure all function calls have any opening and closing parentheses.

```
# file is script.py
# My script to plot the observed visibilities
plotxy('ngc5921.ms', 'uvdist') #yaxis defaults to amplitude
```

This can be done by using the execfile command to execute this script. execfile will execute the script as though you had typed the lines at the CASA prompt.

```
CASA [5]: execfile 'script.py'
-----> execfile('script.py')
```

D.12 How do I exit from CASA?

You can exit CASA by using the `quit` command. This will bring up the query

```
Do you really want to exit ([y]/n)?
```

to give you a chance in case you did not mean to exit. You can also quit using `%exit` or `CTRL-D`.

If you don't want to see the question "Do you really want to exit [y]/n?", then just type `Exit` or `exit` followed by return, and CASA will stop right then and there.

Appendix E

The Measurement Equation and Calibration

The visibilities measured by an interferometer must be calibrated before formation of an image. This is because the wavefronts received and processed by the observational hardware have been corrupted by a variety of effects. These include (but are not exclusive to): the effects of transmission through the atmosphere, the imperfect details amplified electronic (digital) signal and transmission through the signal processing system, and the effects of formation of the cross-power spectra by a correlator. Calibration is the process of reversing these effects to arrive at corrected visibilities which resemble as closely as possible the visibilities that would have been measured in vacuum by a perfect system. The subject of this chapter is the determination of these effects by using the visibility data itself.

E.1 The HBS Measurement Equation

The relationship between the observed and ideal (desired) visibilities on the baseline between antennas i and j may be expressed by the Hamaker-Bregman-Sault *Measurement Equation*¹:

$$\vec{V}_{ij} = J_{ij} \vec{V}_{ij}^{\text{IDEAL}}$$

where \vec{V}_{ij} represents the observed visibility, $\vec{V}_{ij}^{\text{IDEAL}}$ represents the corresponding ideal visibilities, and J_{ij} represents the accumulation of all corruptions affecting baseline ij . The visibilities are indicated as vectors spanning the four correlation combinations which can be formed from dual-polarization signals. These four correlations are related directly to the Stokes parameters which fully describe the radiation. The J_{ij} term is therefore a 4×4 matrix.

Most of the effects contained in J_{ij} (indeed, the most important of them) are antenna-based, i.e., they arise from measurable physical properties of (or above) individual antenna elements in a synthesis array. Thus, adequate calibration of an array of N_{ant} antennas forming $N_{ant}(N_{ant} - 1)/2$ baseline visibilities is usually achieved through the determination of only N_{ant} factors, such that

¹Hamaker, J.P., Bregman, J.D. & Sault, R.J. (1996), *Astronomy and Astrophysics Supplement*, v.117, p.137-147

$J_{ij} = J_i \otimes J_j^*$. For the rest of this chapter, we will usually assume that J_{ij} is factorable in this way, unless otherwise noted.

As implied above, J_{ij} may also be factored into the sequence of specific corrupting effects, each having their own particular (relative) importance and physical origin, which determines their unique algebra. Including the most commonly considered effects, the Measurement Equation can be written:

$$\vec{V}_{ij} = M_{ij} B_{ij} G_{ij} D_{ij} E_{ij} P_{ij} T_{ij} \vec{V}_{ij}^{\text{IDEAL}}$$

where:

- T_{ij} = Polarization-independent multiplicative effects introduced by the troposphere, such as opacity and path-length variation.
- P_{ij} = Parallactic angle, which describes the orientation of the polarization coordinates on the plane of the sky. This term varies according to the type of the antenna mount.
- E_{ij} = Effects introduced by properties of the optical components of the telescopes, such as the collecting area's dependence on elevation.
- D_{ij} = Instrumental polarization response. "D-terms" describe the polarization leakage between feeds (e.g. how much the R-polarized feed picked up L-polarized emission, and vice versa).
- G_{ij} = Electronic gain response due to components in the signal path between the feed and the correlator. This complex gain term G_{ij} includes the scale factor for absolute flux density calibration, and may include phase and amplitude corrections due to changes in the atmosphere (in lieu of T_{ij}). These gains are polarization-dependent.
- B_{ij} = Bandpass (frequency-dependent) response, such as that introduced by spectral filters in the electronic transmission system
- M_{ij} = Baseline-based correlator (non-closing) errors. By definition, these are not factorable into antenna-based parts.

Note that the terms are listed in the order in which they affect the incoming wavefront (G and B represent an arbitrary sequence of such terms depending upon the details of the particular electronic system). Note that M differs from all of the rest in that it is not antenna-based, and thus not factorable into terms for each antenna.

As written above, the measurement equation is very general; not all observations will require treatment of all effects, depending upon the desired dynamic range. E.g., bandpass need only be considered for continuum observations if observed in a channelized mode and very high dynamic range is desired. Similarly, instrumental polarization calibration can usually be omitted when observing (only) total intensity using circular feeds. Ultimately, however, each of these effects occurs at some level, and a complete treatment will yield the most accurate calibration. Modern high-sensitivity instruments such as ALMA and EVLA will likely require a more general calibration

treatment for similar observations with older arrays in order to reach the advertised dynamic ranges on strong sources.

In practice, it is usually far too difficult to adequately measure most calibration effects absolutely (as if in the laboratory) for use in calibration. The effects are usually far too changeable. Instead, the calibration is achieved by making observations of calibrator sources on the appropriate timescales for the relevant effects, and solving the measurement equation for them using the fact that we have $N_{ant}(N_{ant} - 1)/2$ measurements and only N_{ant} factors to determine (except for M which is only sparingly used). (*Note: By partitioning the calibration factors into a series of consecutive effects, it might appear that the number of free parameters is some multiple of N_{ant} , but the relative algebra and timescales of the different effects, as well as the the multiplicity of observed polarizations and channels compensate, and it can be shown that the problem remains well-determined until, perhaps, the effects are direction-dependent within the field of view. Limited solvers for such effects are under study; the `calibrator` tool currently only handles effects which may be assumed constant within the field of view. Corrections for the primary beam are handled in the `imager` tool.*) Once determined, these terms are used to correct the visibilities measured for the scientific target. This procedure is known as cross-calibration (when only phase is considered, it is called phase-referencing).

The best calibrators are point sources at the phase center (constant visibility amplitude, zero phase), with sufficient flux density to determine the calibration factors with adequate SNR on the relevant timescale. The primary gain calibrator must be sufficiently close to the target on the sky so that its observations sample the same atmospheric effects. A bandpass calibrator usually must be sufficiently strong (or observed with sufficient duration) to provide adequate per-channel sensitivity for a useful calibration. In practice, several calibrators are usually observed, each with properties suitable for one or more of the required calibrations.

Synthesis calibration is inherently a bootstrapping process. First, the dominant calibration term is determined, and then, using this result, more subtle effects are solved for, until the full set of required calibration terms is available for application to the target field. The solutions for each successive term are relative to the previous terms. Occasionally, when the several calibration terms are not sufficiently orthogonal, it is useful to re-solve for earlier types using the results for later types, in effect, reducing the effect of the later terms on the solution for earlier ones, and thus better isolating them. This idea is a generalization of the traditional concept of self-calibration, where initial imaging of the target source supplies the visibility model for a re-solve of the gain calibration (G or T). Iteration tends toward convergence to a statistically optimal image. In general, the quality of each calibration and of the source model are mutually dependent. In principle, as long as the solution for any calibration component (or the source model itself) is likely to improve substantially through the use of new information (provided by other improved solutions), it is worthwhile to continue this process.

In practice, these concepts motivate certain patterns of calibration for different types of observation, and the `calibrator` tool in CASA is designed to accommodate these patterns in a general and flexible manner. For a spectral line total intensity observation, the pattern is usually:

1. Solve for G on the bandpass calibrator
2. Solve for B on the bandpass calibrator, using G

3. Solve for G on the primary gain (near-target) and flux density calibrators, using B solutions just obtained
4. Scale G solutions for the primary gain calibrator according to the flux density calibrator solutions
5. Apply G and B solutions to the target data
6. Image the calibrated target data

If opacity and gain curve information are relevant and available, these types are incorporated in each of the steps (in future, an actual solve for opacity from appropriate data may be folded into this process):

1. Solve for G on the bandpass calibrator, using T (opacity) and E (gain curve) solutions already derived.
2. Solve for B on the bandpass calibrator, using G , T (opacity), and E (gain curve) solutions.
3. Solve for G on primary gain (near-target) and flux density calibrators, using B , T (opacity), and E (gain curve) solutions.
4. Scale G solutions for the primary gain calibrator according to the flux density calibrator solutions
5. Apply T (opacity), E (gain curve), G , and B solutions to the target data
6. Image the calibrated target data

For continuum polarimetry, the typical pattern is:

1. Solve for G on the polarization calibrator, using (analytical) P solutions.
2. Solve for D on the polarization calibrator, using P and G solutions.
3. Solve for G on primary gain and flux density calibrators, using P and D solutions.
4. Scale G solutions for the primary gain calibrator according to the flux density calibrator solutions.
5. Apply P , D , and G solutions to target data.
6. Image the calibrated target data.

For a spectro-polarimetry observation, these two examples would be folded together.

In all cases the calibrator model must be adequate at each solve step. At high dynamic range and/or high resolution, many calibrators which are nominally assumed to be point sources become slightly resolved. If this has biased the calibration solutions, the offending calibrator may be imaged at any point in the process and the resulting model used to improve the calibration. Finally, if sufficiently strong, the target may be self-calibrated as well.

E.2 General Calibrator Mechanics

The calibrator tasks/tool are designed to solve and apply solutions for all of the solution types listed above (and more are in the works). This leads to a single basic sequence of execution for all solves, regardless of type:

1. Set the calibrator model visibilities
2. Select the visibility data which will be used to solve for a calibration type
3. Arrange to apply any already-known calibration types (the first time through, none may yet be available)
4. Arrange to solve for a specific calibration type, including specification of the solution timescale and other specifics
5. Execute the solve process
6. Repeat 1-4 for all required types, using each result, as it becomes available, in step 2, and perhaps repeating for some types to improve the solutions

By itself, this sequence doesn't guarantee success; the data provided for the solve must have sufficient SNR on the appropriate timescale, and must provide sufficient leverage for the solution (e.g., D solutions require data taken over a sufficient range of parallactic angle in order to separate the source polarization contribution from the instrumental polarization).

Appendix F

Annotated Example Scripts

Note: These data sets are available with the full CASA rpm distribution. Other data sets can be made available upon request. The scripts are intended to illustrate the types of commands needed for different types of reduction/astronomical observations.

ALERT: Some tasks are still under development and we will be occasionally updating the syntax of the tasks, which may break older versions of the scripts. You can find the latest versions of these (and other) scripts at:

<http://casa.nrao.edu/Doc/Scripts/>

We also provide a number of annotated data reduction tutorials and scripts on our wiki page

<http://casaguides.nrao.edu>

Those will cover more recent data such as EVLA OSRO data.

F.1 NGC 5921 — VLA red-shifted HI emission

This script demonstrates basic spectral calibration and imaging, but does not include any self-calibration steps.

The latest version of this script can be found at:

http://casa.nrao.edu/Doc/Scripts/ngc5921_demo.py

```
#####  
#                                                                 #  
# Use Case Script for NGC 5921                                   #  
#                                                                 #
```

```

# Converted by STM 2007-05-26 #
# Updated STM 2007-06-15 (Alpha Patch 1) #
# Updated STM 2007-09-05 (Alpha Patch 2+) #
# Updated STM 2007-09-18 (Alpha Patch 2+) #
# Updated STM 2007-09-18 (Pre-Beta) add immoments #
# Updated STM 2007-10-04 (Beta) update #
# Updated STM 2007-10-10 (Beta) add export #
# Updated STM 2007-11-08 (Beta Patch 0.5) add RRusk stuff #
# Updated STM 2008-03-25 (Beta Patch 1.0) #
# Updated STM 2008-05-23 (Beta Patch 2.0) new tasking/clean/cal #
# Updated STM 2008-06-11 (Beta Patch 2.0) #
# Updated STM 2008-06-13 (Beta Patch 2.0) demo version #
# Updated STM 2008-06-14 (Beta Patch 2.0) post-school update #
# Updated STM 2008-07-06 (Beta Patch 2.0) regression version #
# Updated STM 2009-05-26 (Beta Patch 4.0) McMaster demo #
# Updated STM 2009-12-02 (Release 0) #
# Revised JO 2010-04-13 (Release 3.0.1) edit viewer call #
# #
# Features Tested: #
# The script illustrates end-to-end processing with CASA #
# as depicted in the following flow-chart. #
# #
# Filenames will have the <prefix> = 'ngc5921.demo' #
# #
# Input Data Process Output Data #
# #
# NGC5921.fits --> importuvfits --> <prefix>.ms + #
# (1.4GHz, | <prefix>.ms.flagversions #
# 63 sp chan, v #
# D-array) listobs --> casapy.log #
# | #
# v #
# flagautocorr #
# | #
# v #
# setjy #
# | #
# v #
# bandpass --> <prefix>.bcal #
# | #
# v #
# gaincal --> <prefix>.gcal #
# | #
# v #
# fluxscale --> <prefix>.fluxscale #
# | #
# v #
# applycal --> <prefix>.ms #
# | #
# v #
# split --> <prefix>.cal.split.ms #
# #

```

```

#           |           #
#           v           #
#           split      --> <prefix>.src.split.ms      #
#           |           #
#           v           #
#           exportuvfits --> <prefix>.split.uvfits    #
#           |           #
#           v           #
#           uvcontsub  --> <prefix>.ms.cont +         #
#           |           <prefix>.ms.contsub          #
#           v           #
#           clean      --> <prefix>.clean.image +     #
#           |           <prefix>.clean.model +       #
#           |           <prefix>.clean.residual      #
#           v           #
#           exportfits --> <prefix>.clean.fits        #
#           |           #
#           v           #
#           imhead     --> casapy.log                 #
#           |           #
#           v           #
#           imstat     --> xstat (parameter)          #
#           |           #
#           v           #
#           immoments  --> <prefix>.moments.integrated + #
#           |           <prefix>.moments.weighted_coord #
#           v           #
#####
print 'Demo Script for NGC5921 VLA HI observation'
print 'Version for Release 0 (3.0.0) 2-Dec-2009'
print ''

import time
import os

scriptmode = True
#
# The prefix to use for all output files
prefix='ngc5921.demo'

# Set up some useful variables (these will also be set later on)
msfile = prefix + '.ms'
btable = prefix + '.bcal'
gtable = prefix + '.gcal'
ftable = prefix + '.fluxscale'
splitms = prefix + '.src.split.ms'
inname = prefix + '.cleaning'

#
# Get to path to the CASA home and strip off the name
pathname=os.environ.get('CASAPATH').split()[0]

```

```

# This is where the NGC5921 UVFITS data will be
fitsdata=pathname+'/data/demo/NGC5921.fits'
#
# Or uncomment the following to use data in current directory
#fitsdata='NGC5921.fits'

# Clean up old files
# Use rmtables on ms and cal tables to clear cache
# (not working on multiple runs for 2.4.0 release)
#rmtables(msfile)
#rmtables(btable)
#rmtables(gtable)
#rmtables(ftable)
#rmtables(ftable)
#rmtables(splitms+'*')
#rmtables(imname+'.*')
#rmtables(prefix+'.moments*')

# Final clean up of auxiliary files
os.system('rm -rf '+prefix+'*')

#
#=====
#
# Import the data from FITS to MS
#
print '--Import--'

# Safest to start from task defaults
default('importuvfits')

# Set up the MS filename and save as new global variable
msfile = prefix + '.ms'

# Use task importuvfits
fitsfile = fitsdata
vis = msfile

saveinputs('importuvfits',prefix+'.importuvfits.saved')

importuvfits()

#
# Note that there will be a ngc5921.demo.ms.flagversions
# there containing the initial flags as backup for the main ms
# flags.
#
#=====
#
# List a summary of the MS

```

```

#
print '--Listobs--'

# Don't default this one and make use of the previous setting of
# vis. Remember, the variables are GLOBAL!

# You may wish to see more detailed information, like the scans.
# In this case use the verbose = True option
verbose = True

listobs()

# You should get in your logger window and in the casapy.log file
# something like:
#
# MeasurementSet Name: /home/sandrock2/smyers/Testing2/Sep07/ngc5921.demo.ms
# MS Version 2
#
# Observer: TEST      Project:
# Observation: VLA
#
# Data records: 22653      Total integration time = 5280 seconds
#   Observed from 09:19:00 to 10:47:00
#
#   ObservationID = 0      ArrayID = 0
#   Date           Timerange           Scan  FldId FieldName           SpwIds
#   13-Apr-1995/09:19:00.0 - 09:24:30.0    1     0 1331+30500002_0 [0]
#                   09:27:30.0 - 09:29:30.0    2     1 1445+09900002_0 [0]
#                   09:33:00.0 - 09:48:00.0    3     2 N5921_2          [0]
#                   09:50:30.0 - 09:51:00.0    4     1 1445+09900002_0 [0]
#                   10:22:00.0 - 10:23:00.0    5     1 1445+09900002_0 [0]
#                   10:26:00.0 - 10:43:00.0    6     2 N5921_2          [0]
#                   10:45:30.0 - 10:47:00.0    7     1 1445+09900002_0 [0]
#
# Fields: 3
#   ID  Code Name           Right Ascension  Declination     Epoch
#   0   C   1331+30500002_013:31:08.29  +30.30.32.96  J2000
#   1   A   1445+09900002_014:45:16.47  +09.58.36.07  J2000
#   2           N5921_2           15:22:00.00    +05.04.00.00  J2000
#
# Spectral Windows: (1 unique spectral windows and 1 unique polarization setups)
# SpwID #Chans Frame Ch1(MHz)  Resoln(kHz) TotBW(kHz) Ref(MHz) Corrs
# 0      63 LSRK 1412.68608 24.4140625 1550.19688 1413.44902 RR LL
#
# Feeds: 28: printing first row only
# Antenna Spectral Window # Receptors Polarizations
# 1 -1 2 [ R, L]
#
# Antennas: 27:
# ID Name Station Diam. Long. Lat.
# 0 1 VLA:N7 25.0 m -107.37.07.2 +33.54.12.9

```

```

# 1 2 VLA:W1 25.0 m -107.37.05.9 +33.54.00.5
# 2 3 VLA:W2 25.0 m -107.37.07.4 +33.54.00.9
# 3 4 VLA:E1 25.0 m -107.37.05.7 +33.53.59.2
# 4 5 VLA:E3 25.0 m -107.37.02.8 +33.54.00.5
# 5 6 VLA:E9 25.0 m -107.36.45.1 +33.53.53.6
# 6 7 VLA:E6 25.0 m -107.36.55.6 +33.53.57.7
# 7 8 VLA:W8 25.0 m -107.37.21.6 +33.53.53.0
# 8 9 VLA:N5 25.0 m -107.37.06.7 +33.54.08.0
# 9 10 VLA:W3 25.0 m -107.37.08.9 +33.54.00.1
# 10 11 VLA:N4 25.0 m -107.37.06.5 +33.54.06.1
# 11 12 VLA:W5 25.0 m -107.37.13.0 +33.53.57.8
# 12 13 VLA:N3 25.0 m -107.37.06.3 +33.54.04.8
# 13 14 VLA:N1 25.0 m -107.37.06.0 +33.54.01.8
# 14 15 VLA:N2 25.0 m -107.37.06.2 +33.54.03.5
# 15 16 VLA:E7 25.0 m -107.36.52.4 +33.53.56.5
# 16 17 VLA:E8 25.0 m -107.36.48.9 +33.53.55.1
# 17 18 VLA:W4 25.0 m -107.37.10.8 +33.53.59.1
# 18 19 VLA:E5 25.0 m -107.36.58.4 +33.53.58.8
# 19 20 VLA:W9 25.0 m -107.37.25.1 +33.53.51.0
# 20 21 VLA:W6 25.0 m -107.37.15.6 +33.53.56.4
# 21 22 VLA:E4 25.0 m -107.37.00.8 +33.53.59.7
# 23 24 VLA:E2 25.0 m -107.37.04.4 +33.54.01.1
# 24 25 VLA:N6 25.0 m -107.37.06.9 +33.54.10.3
# 25 26 VLA:N9 25.0 m -107.37.07.8 +33.54.19.0
# 26 27 VLA:N8 25.0 m -107.37.07.5 +33.54.15.8
# 27 28 VLA:W7 25.0 m -107.37.18.4 +33.53.54.8

```

#

Tables:

```

# MAIN 22653 rows
# ANTENNA 28 rows
# DATA_DESCRIPTION 1 row
# DOPPLER <absent>
# FEED 28 rows
# FIELD 3 rows
# FLAG_CMD <empty>
# FREQ_OFFSET <absent>
# HISTORY 273 rows
# OBSERVATION 1 row
# POINTING 168 rows
# POLARIZATION 1 row
# PROCESSOR <empty>
# SOURCE 3 rows
# SPECTRAL_WINDOW 1 row
# STATE <empty>
# SYSCAL <absent>
# WEATHER <absent>

```

#

#

#=====

#

Get rid of the autocorrelations from the MS


```

#
print '--Flagautocorr--'

# Don't default this one either, there is only one parameter (vis)

flagautocorr()

#
#=====
#
# Set the fluxes of the primary calibrator(s)
#
print '--Setjy--'
default('setjy')

vis = msfile

#
# 1331+305 = 3C286 is our primary calibrator
# Use the wildcard on the end of the source name
# since the field names in the MS have inherited the
# AIPS qualifiers
field = '1331+305*'

# This is 1.4GHz D-config and 1331+305 is sufficiently unresolved
# that we dont need a model image. For higher frequencies
# (particularly in A and B config) you would want to use one.
modimage = ''

# Setjy knows about this source so we dont need anything more

saveinputs('setjy',prefix+'.setjy.saved')

# Pause script if you are running in scriptmode
if scriptmode:
    inp()
    user_check=raw_input('Return to continue script\n')

setjy()

#
# You should see something like this in the logger and casapy.log file:
#
# 1331+30500002_0 spwid= 0 [I=14.76, Q=0, U=0, V=0] Jy, (Perley-Taylor 99)
#
# So its using 14.76Jy as the flux of 1331+305 in the single Spectral Window
# in this MS.
#
#=====
#
# Bandpass calibration

```

```

#
print '--Bandpass--'
default('bandpass')

# We can first do the bandpass on the single 5min scan on 1331+305
# At 1.4GHz phase stability should be sufficient to do this without
# a first (rough) gain calibration. This will give us the relative
# antenna gain as a function of frequency.

vis = msfile

# set the name for the output bandpass caltable
btable = prefix + '.bcal'
caltable = btable

# No gain tables yet
gaintable = ''
gainfield = ''
interp = ''

# Use flux calibrator 1331+305 = 3C286 (FIELD_ID 0) as bandpass calibrator
field = '0'
# all channels
spw = ''
# No other selection
selectdata = False

# In this band we do not need a-priori corrections for
# antenna gain-elevation curve or atmospheric opacity
# (at 8GHz and above you would want these)
gaincurve = False
opacity = 0.0

# Choose bandpass solution type
# Pick standard time-binned B (rather than BPOLY)
bandtype = 'B'

# set solution interval arbitrarily long (get single bpass)
solint = 'inf'
combine = 'scan'

# reference antenna Name 15 (15=VLA:N2) (Id 14)
refant = '15'

saveinputs('bandpass',prefix+'.bandpass.saved')

# Pause script if you are running in scriptmode
if scriptmode:
    inp()
    user_check=raw_input('Return to continue script\n')

```

```

bandpass()

#
#=====
#
# Use plotcal to examine the bandpass solutions
#
print '--Plotcal (bandpass)--'
default('plotcal')

caltable = btable
field = '0'

# Set up 2x1 panels - upper panel amp vs. channel
subplot = 211
yaxis = 'amp'
# No output file yet (wait to plot next panel)

saveinputs('plotcal',prefix+'.plotcal.b.amp.saved')

if scriptmode:
    showgui = True
else:
    showgui = False

plotcal()
#
# Set up 2x1 panels - lower panel phase vs. channel
subplot = 212
yaxis = 'phase'

saveinputs('plotcal',prefix+'.plotcal.b.phase.saved')

#
# Note the rolloff in the start and end channels. Looks like
# channels 6-56 (out of 0-62) are the best

# Pause script if you are running in scriptmode
if scriptmode:
    # If you want to do this interactively and iterate over antenna, set
    # iteration = 'antenna'
    showgui = True
    plotcal()
    user_check=raw_input('Return to continue script\n')
else:
    # No GUI for this script
    showgui = False
    # Now send final plot to file in PNG format (via .png suffix)
    figfile = caltable + '.plotcal.png'
    plotcal()

```

```

=====
#
# Gain calibration
#
print '--Gaincal--'
default('gaincal')

# Armed with the bandpass, we now solve for the
# time-dependent antenna gains

vis = msfile

# set the name for the output gain caltable
gtable = prefix + '.gcal'
caltable = gtable

# Use our previously determined bandpass
# Note this will automatically be applied to all sources
# not just the one used to determine the bandpass
gaintable = btable
gainfield = ''

# Use nearest (there is only one bandpass entry)
interp = 'nearest'

# Gain calibrators are 1331+305 and 1445+099 (FIELD_ID 0 and 1)
field = '0,1'

# We have only a single spectral window (SPW 0)
# Choose 51 channels 6-56 out of the 63
# to avoid end effects.
# Channel selection is done inside spw
spw = '0:6~56'

# No other selection
selectdata = False

# In this band we do not need a-priori corrections for
# antenna gain-elevation curve or atmospheric opacity
# (at 8GHz and above you would want these)
gaincurve = False
opacity = 0.0

# scan-based G solutions for both amplitude and phase
gaintype = 'G'
solint = 'inf'
combine = ''
calmode = 'ap'

# minimum SNR allowed
minsnr = 1.0

```

```

# reference antenna 15 (15=VLA:N2)
refant = '15'

saveinputs('gaincal',prefix+'.gaincal.saved')

if scriptmode:
    inp()
    user_check=raw_input('Return to continue script\n')

gaincal()

#
#=====
#
# Bootstrap flux scale
#
print '--Fluxscale--'
default('fluxscale')

vis = msfile

# set the name for the output rescaled caltable
ftable = prefix + '.fluxscale'
fluxtable = ftable

# point to our first gain cal table
caltable = gtable

# we will be using 1331+305 (the source we did setjy on) as
# our flux standard reference - note its extended name as in
# the FIELD table summary above (it has a VLA seq number appended)
reference = '1331*'

# we want to transfer the flux to our other gain cal source 1445+099
transfer = '1445*'

saveinputs('fluxscale',prefix+'.fluxscale.saved')

# Pause script if you are running in scriptmode
if scriptmode:
    inp()
    user_check=raw_input('Return to continue script\n')

fluxscale()

# In the logger you should see something like:
# Flux density for 1445+09900002_0 in SpW=0 is:
#      2.48576 +/- 0.00123122 (SNR = 2018.94, nAnt= 27)

# If you run plotcal() on the tablein = 'ngc5921.demo.fluxscale'

```

```

# you will see now it has brought the amplitudes in line between
# the first scan on 1331+305 and the others on 1445+099

#
#=====
#
# Now use plotcal to examine the gain solutions
#
print '--Plotcal (fluxscaled gains)--'
default('plotcal')

caltable = ftable
field = '0,1'

# Set up 2x1 panels - upper panel amp vs. time
subplot = 211
yaxis = 'amp'
# No output file yet (wait to plot next panel)

saveinputs('plotcal',prefix+'.plotcal.gscaled.amp.saved')

if scriptmode:
    showgui = True
else:
    showgui = False

plotcal()
#
# Set up 2x1 panels - lower panel phase vs. time
subplot = 212
yaxis = 'phase'

saveinputs('plotcal',prefix+'.plotcal.gscaled.phase.saved')

#
# The amp and phase coherence looks good

# Pause script if you are running in scriptmode
if scriptmode:
    # If you want to do this interactively and iterate over antenna, set
    #iteration = 'antenna'
    showgui = True
    plotcal()
    user_check=raw_input('Return to continue script\n')
else:
    # No GUI for this script
    showgui = False
    # Now send final plot to file in PNG format (via .png suffix)
    figfile = caltable + '.plotcal.png'
    plotcal()

```

```

=====
#
# Apply our calibration solutions to the data
# (This will put calibrated data into the CORRECTED_DATA column)
#
print '--ApplyCal--'
default('applycal')

vis = msfile

# We want to correct the calibrators using themselves
# and transfer from 1445+099 to itself and the target N5921

# Start with the fluxscale/gain and bandpass tables
gaintable = [ftable,btable]

# pick the 1445+099 out of the gain table for transfer
# use all of the bandpass table
gainfield = ['1','*']

# interpolation using linear for gain, nearest for bandpass
interp = ['linear','nearest']

# only one spw, do not need mapping
spwmap = []

# all channels
spw = ''
selectdata = False

# as before
gaincurve = False
opacity = 0.0

# select the fields for 1445+099 and N5921
field = '1,2'

applycal()

# Now for completeness apply 1331+305 to itself

field = '0'
gainfield = ['0','*']

# The CORRECTED_DATA column now contains the calibrated visibilities

saveinputs('applycal',prefix+'.applycal.saved')

# Pause script if you are running in scriptmode
if scriptmode:
    inp()

```

```

    user_check=raw_input('Return to continue script\n')

applycal()

#
#=====
#
# Now use plotxy to plot the calibrated target data (before contsub)
#
print '--Plotxy (NGC5921)--'
default('plotxy')

vis = msfile

field = '2'
# Edge channels are bad
spw = '0:4`59'

# Time average across scans
timebin = '86000.'
crossscans = True

# Set up 2x1 panels - upper panel amp vs. channel
subplot = 211
xaxis = 'channel'
yaxis = 'amp'
datacolumn = 'corrected'
# No output file yet (wait to plot next panel)

saveinputs('plotxy',prefix+'.plotxy.final.amp.saved')

print "Amp averaged across time and baseline (upper)"

figfile = ''
if scriptmode:
    interactive = True
else:
    interactive = False

plotxy()
#
# Set up 2x1 panels - lower panel phase vs. time
subplot = 212
yaxis = 'phase'
datacolumn = 'corrected'
# Time average across scans and baselines
timebin = '86000.'
crossscans = True
crossbls = True

saveinputs('plotxy',prefix+'.plotxy.final.phase.saved')

```



```

print "Phase averaged across time and baseline (lower)"

print "Final calibrated data"

# Pause script if you are running in scriptmode
if scriptmode:
    interactive = True
    figfile = ''
    plotxy()
    user_check=raw_input('Return to continue script\n')
else:
    interactive = False
    # Now send final plot to file in PNG format (via .png suffix)
    figfile = vis + '.plotxy.png'
    plotxy()

#=====
#
# Split the sources out, pick off the CORRECTED_DATA column
#
#
# Split NGC5921 data (before continuum subtraction)
#
print '--Split NGC5921 Data--'
default('split')

vis = msfile
splitms = prefix + '.src.split.ms'
outputvis = splitms
field = 'N5921*'
spw = ''
datacolumn = 'corrected'

saveinputs('split',prefix+'.split.n5921.saved')

split()

print "Created "+splitms

# If you want, split out the calibrator 1445+099 field, all chans
#print '--Split 1445+099 Data--'
#
#calsplitms = prefix + '.cal.split.ms'
#outputvis = calsplitms
#field = '1445*'
#
#saveinputs('split',prefix+'.split.1445.saved')
#
#split()

```

```

=====
#
# Here is how to export the NGC5921 data as UVFITS
# Start with the split file.
# Since this is a split dataset, the calibrated data is
# in the DATA column already.
# Write as a multisource UVFITS (with SU table)
# even though it will have only one field in it
# Run asynchronously so as not to interfere with other tasks
# (BETA: also avoids crash on next importuvfits)
#
#print '--Export UVFITS--'
#default('exportuvfits')
#
#srcuvfits = prefix + '.split.uvfits'
#
#vis = splitms
#fitsfile = srcuvfits
#datacolumn = 'data'
#multisource = True
#async = True
#
#saveinputs('exportuvfits',prefix+'.exportuvfits.saved')
#
#myhandle = exportuvfits()
#
#print "The return value for this exportuvfits async task for tm is "+str(myhandle)

=====
#
# UV-plane continuum subtraction on the target
# use the split ms
# (this will update the CORRECTED_DATA column)
#
print '--UV Continuum Subtract--'
default('uvcontsub')

vis = splitms

field = 'N5921*'
# Use channels 4-6 and 50-59 for continuum
fitspw='0:4~6;50~59'

# Output all of spw 0
spw = '0'

# Averaging time (none)
solint = 0.0

# Fit only a mean level

```

```

fitorder = 0

# Do the uv-plane subtraction
fitmode = 'subtract'

# Let it split out the data automatically for us
splitdata = True

saveinputs('uvcontsub',prefix+'.uvcontsub.saved')

# Pause script if you are running in scriptmode
if scriptmode:
    inp()
    user_check=raw_input('Return to continue script\n')

uvcontsub()

# You will see it made two new MS:
# <vis>.cont
# <vis>.contsub

srcsplitms = splitms + '.contsub'

# Note that ngc5921.demo.ms.contsub contains the uv-subtracted
# visibilities (in its DATA column), and ngc5921.demo.ms.cont
# the pseudo-continuum visibilities (as fit).

# The original ngc5921.demo.ms now contains the uv-continuum
# subtracted vis in its CORRECTED_DATA column and the continuum
# in its MODEL_DATA column as per the fitmode='subtract'

# Done with calibration
#=====
#
# Here is how to make a dirty image cube
#
#print '--Clean (dirty image)--'
#default('clean')

# Pick up our split source continuum-subtracted data
#vis = srcsplitms
#dirtyname = prefix + '.dirtyimg'
#imagename = dirtyname
#
#mode = 'channel'
#nchan = 46
#start = 5
#width = 1
#
#field = '0'
#spw = ''

```

```

#imsize = [256,256]
#cell = [15.,15.]
#weighting = 'briggs'
#robust = 0.5

# No cleaning
#niter = 0

#saveinputs('clean',prefix+'.invert.saved')

# Pause script if you are running in scriptmode
#if scriptmode:
#   inp()
#   user_check=raw_input('Return to continue script\n')
#
#clean()

#dirtyimage = dirtyname+'.image'

# Get the dirty image cube statistics
#dirtystats = imstat(dirtyimage)

#=====
#
# Now clean an image cube of N5921
#
print '--Clean (clean)--'
default('clean')

# Pick up our split source continuum-subtracted data
vis = srcsplitms

# Make an image root file name
inname = prefix + '.cleanimg'
imagenname = inname

# Set up the output image cube
mode = 'channel'
nchan = 46
start = 5
width = 1

# This is a single-source MS with one spw
field = '0'
spw = ''

# Standard gain factor 0.1
gain = 0.1

# Set the output image size and cell size (arcsec)
imsize = [256,256]

```

```
# Do a simple Clark clean
psfmode = 'clark'
# No Cotton-Schwab iterations
csclean = False

# If desired, you can do a Cotton-Schwab clean
# but will have only marginal improvement for this data
#csclean = True
# Twice as big for Cotton-Schwab (cleans inner quarter)
#imsize = [512,512]

# Pixel size 15 arcsec for this data (1/3 of 45" beam)
# VLA D-config L-band
cell = [15.,15.]

# Fix maximum number of iterations
niter = 6000

# Also set flux residual threshold (in mJy)
threshold=8.0

# Set up the weighting
# Use Briggs weighting (a moderate value, on the uniform side)
weighting = 'briggs'
robust = 0.5

# Set a cleanbox +/-20 pixels around the center 128,128
mask = [108,108,148,148]

# But if you had a cleanbox saved in a file, e.g. "regionfile.txt"
# you could use it:
#mask='regionfile.txt'
#
# If you don't want any clean boxes or masks, then
#mask = ''

# If you want interactive clean set to True
#interactive=True
interactive=False

saveinputs('clean',prefix+'.clean.saved')

# Pause script if you are running in scriptmode
if scriptmode:
    inp()
    user_check=raw_input('Return to continue script\n')

clean()

# Should find stuff in the logger like:
```

```

#
# Fitted beam used in restoration: 51.5643 by 45.6021 (arcsec)
#   at pa 14.5411 (deg)
#
# It will have made the images:
# -----
# ngc5921.demo.cleanimg.flux
# ngc5921.demo.cleanimg.image
# ngc5921.demo.cleanimg.mask
# ngc5921.demo.cleanimg.model
# ngc5921.demo.cleanimg.psf
# ngc5921.demo.cleanimg.residual

clnimage = imname+'.image'

=====
#
# Done with imaging
# Now view the image cube of N5921
#
if scriptmode:
    print '--View image--'
    print "Use Spectral Profile Tool to get line profile in box in center"
    viewer(clnimage)
    user_check=raw_input('Return to continue script\n')

=====
#
# Here is how to export the Final CLEAN Image as FITS
# Run asynchronously so as not to interfere with other tasks
# (BETA: also avoids crash on next importfits)
#
#print '--Final Export CLEAN FITS--'
#default('exportfits')
#
#clnfits = prefix + '.cleanimg.fits'
#
#imagenname = clnimage
#fitsimage = clnfits
#async = True
#
#saveinputs('exportfits',prefix+'.exportfits.saved')
#
#myhandle2 = exportfits()
#
#print "The return value for this exportfits async task for tm is "+str(myhandle2)

=====
#
# Print the image header
#

```

```

print '--Imhead--'
default('imhead')

imagenname = clnimage

mode = 'summary'

imhead()

# A summary of the cube will be seen in the logger

#=====
#
# Get the cube statistics
#
print '--Imstat (cube)--'
default('imstat')

imagenname = clnimage

# Do whole image
box = ''
# or you could stick to the cleanbox
#box = '108,108,148,148'

cubestats = imstat()

# Statistics will printed to the terminal, and the output
# parameter will contain a dictionary of the statistics

#=====
#
# Get some image moments
#
print '--ImMoments--'
default('immoments')

imagenname = clnimage

# Do first and second moments
moments = [0,1]

# Need to mask out noisy pixels, currently done
# using hard global limits
excludepix = [-100,0.009]

# Collapse along the spectral (channel) axis
axis = 'spectral'
# Include all planes
chans = ''

```

```

# Output root name
momfile = prefix + '.moments'
outfile = momfile

saveinputs('immoments',prefix+'.immoments.saved')

# Pause script if you are running in scriptmode
if scriptmode:
    inp()
    user_check=raw_input('Return to continue script\n')

immoments()

momzeroimage = momfile + '.integrated'
momoneimage = momfile + '.weighted_coord'

# It will have made the images:
# -----
# ngc5921.demo.moments.integrated
# ngc5921.demo.moments.weighted_coord

#
#=====
#
# Get some statistics of the moment images
#
print '--Imstat (moments)--'
default('imstat')

imagename = momzeroimage
momzerostats = imstat()

imagename = momoneimage
momonestats = imstat()

#=====
#
# Now view the moments
#
if scriptmode:
    print '--View image (Moments)--'
    viewer(momzeroimage)
    print "You can add mom-1 image "+momoneimage
    print "as a contour plot"
    user_check=raw_input('Return to continue script\n')

#=====
#
# Set up an output logfile
import datetime
datestring=datetime.datetime.isoformat(datetime.datetime.today())

```



```

outfile = 'out.'+prefix+'.'+datestring+'.log'
logfile=open(outfile,'w')
print >>logfile,'Results for '+prefix+' :'
print >>logfile,""

#=====
#
# Can do some image statistics if you wish
# Treat this like a regression script
# WARNING: currently requires toolkit
#
print ' NGC5921 results '
print ' ===== '

print >>logfile,' NGC5921 results '
print >>logfile,' ===== '

#
# Use the ms tool to get max of the MSs
# Eventually should be available from a task
#
# Pull the max cal amp value out of the MS (if you split this)
#ms.open(calsplitms)
#thistest_cal = max(ms.range(["amplitude"]).get('amplitude'))
#ms.close()
#oldtest_cal = 34.0338668823
#diff_cal = abs((oldtest_cal-thistest_cal)/oldtest_cal)
#
#print ' Calibrator data ampl max = ',thistest_cal
#print ' Previous: cal data max = ',oldtest_cal
#print ' Difference (fractional) = ',diff_cal
#print ''
#
#print >>logfile,' Calibrator data ampl max = ',thistest_cal
#print >>logfile,' Previous: cal data max = ',oldtest_cal
#print >>logfile,' Difference (fractional) = ',diff_cal
#print >>logfile,''

# Pull the max src amp value out of the MS
ms.open(srccsplitms)
thistest_src = max(ms.range(["amplitude"]).get('amplitude'))
ms.close()
oldtest_src = 46.2060050964 # now in all chans
diff_src = abs((oldtest_src-thistest_src)/oldtest_src)

print ' Target Src data ampl max = ',thistest_src
print ' Previous: src data max = ',oldtest_src
print ' Difference (fractional) = ',diff_src
print ''

```

```

print >>logfile,' Target Src data ampl max = ',thistest_src
print >>logfile,' Previous: src data max = ',oldtest_src
print >>logfile,' Difference (fractional) = ',diff_src
print >>logfile,''

#
# Now use the stats produced by imstat above
#
# DIRTY IMAGE MAX & RMS (IF YOU MADE A DIRTY IMAGE)
#
#thistest_dirtymax=dirtystats['max'][0]
#oldtest_dirtymax = 0.0515365377069
#diff_dirtymax = abs((oldtest_dirtymax-thistest_dirtymax)/oldtest_dirtymax)
#
#print ' Dirty image max = ',thistest_dirtymax
#print ' Previous: max = ',oldtest_dirtymax
#print ' Difference (fractional) = ',diff_dirtymax
#print ''
#
#print >>logfile,' Dirty Image max = ',thistest_dirtymax
#print >>logfile,' Previous: max = ',oldtest_dirtymax
#print >>logfile,' Difference (fractional) = ',diff_dirtymax
#print >>logfile,''
#
#thistest_dirtyrms=dirtystats['rms'][0]
#oldtest_dirtyrms = 0.00243866862729
#diff_dirtyrms = abs((oldtest_dirtyrms-thistest_dirtyrms)/oldtest_dirtyrms)
#
#print ' Dirty image rms = ',thistest_dirtyrms
#print ' Previous: rms = ',oldtest_dirtyrms
#print ' Difference (fractional) = ',diff_dirtyrms
#print ''
#
#print >>logfile,' Dirty Image rms = ',thistest_dirtyrms
#print >>logfile,' Previous: rms = ',oldtest_dirtyrms
#print >>logfile,' Difference (fractional) = ',diff_dirtyrms
#print >>logfile,''

# Now the clean image
#
thistest_immax=cubestats['max'][0]
oldtest_immax = 0.052414759993553162
diff_immax = abs((oldtest_immax-thistest_immax)/oldtest_immax)

print ' Clean image max = ',thistest_immax
print ' Previous: max = ',oldtest_immax
print ' Difference (fractional) = ',diff_immax
print ''

print >>logfile,' Clean Image max = ',thistest_immax
print >>logfile,' Previous: max = ',oldtest_immax

```

```

print >>logfile,' Difference (fractional) = ',diff_immax
print >>logfile,''

thistest_imrms=cubestats['rms'][0]
oldtest_imrms = 0.0020218724384903908
diff_imrms = abs((oldtest_imrms-thistest_imrms)/oldtest_imrms)

print ' Clean image rms = ',thistest_imrms
print ' Previous: rms = ',oldtest_imrms
print ' Difference (fractional) = ',diff_imrms
print ''

print >>logfile,' Clean image rms = ',thistest_imrms
print >>logfile,' Previous: rms = ',oldtest_imrms
print >>logfile,' Difference (fractional) = ',diff_imrms
print >>logfile,''

# Now the moment images
#
thistest_momzeromax=momzerostats['max'][0]
oldtest_momzeromax = 1.40223777294
diff_momzeromax = abs((oldtest_momzeromax-thistest_momzeromax)/oldtest_momzeromax)

print ' Moment 0 image max = ',thistest_momzeromax
print ' Previous: m0 max = ',oldtest_momzeromax
print ' Difference (fractional) = ',diff_momzeromax
print ''

print >>logfile,' Moment 0 image max = ',thistest_momzeromax
print >>logfile,' Previous: m0 max = ',oldtest_momzeromax
print >>logfile,' Difference (fractional) = ',diff_momzeromax
print >>logfile,''

thistest_momoneavg=momonestats['mean'][0]
oldtest_momoneavg = 1479.77119646
diff_momoneavg = abs((oldtest_momoneavg-thistest_momoneavg)/oldtest_momoneavg)

print ' Moment 1 image mean = ',thistest_momoneavg
print ' Previous: m1 mean = ',oldtest_momoneavg
print ' Difference (fractional) = ',diff_momoneavg
print ''
print '--- Done ---'

print >>logfile,' Moment 1 image mean = ',thistest_momoneavg
print >>logfile,' Previous: m1 mean = ',oldtest_momoneavg
print >>logfile,' Difference (fractional) = ',diff_momoneavg
print >>logfile,''
print >>logfile,'--- Done ---'

# Should see output like:
#

```

```

# Clean image max should be 0.0524147599936
# Found : Image Max = 0.0523551553488
# Difference (fractional) = 0.00113717290288
#
# Clean image rms should be 0.00202187243849
# Found : Image rms = 0.00202226242982
# Difference (fractional) = 0.00019288621809
#
# Moment 0 image max should be 1.40223777294
# Found : Moment 0 Max = 1.40230333805
# Difference (fractional) = 4.67574844349e-05
#
# Moment 1 image mean should be 1479.77119646
# Found : Moment 1 Mean = 1479.66974528
# Difference (fractional) = 6.85586935973e-05
#
#=====
# Done
#
logfile.close()
print "Results are in "+outfile

```

F.2 Jupiter — VLA continuum polarization

This script demonstrates continuum polarization calibration and imaging, including self-calibration. There is also extensive interactive flagging, and image analysis.

The latest version of this script can be found at:

http://casa.nrao.edu/Doc/Scripts/jupiter6cm_demo.py

```

#####
#                                                                 #
# Use Case Script for Jupiter 6cm VLA                            #
# Trimmed down from Use Case jupiter6cm_usecase.py             #
#                                                                 #
# Updated STM 2008-05-15 (Beta Patch 2.0)                       #
# Updated STM 2008-06-11 (Beta Patch 2.0)                       #
# Updated STM 2008-06-12 (Beta Patch 2.0) for summer school demo #
# Updated STM 2008-06-13 (Beta Patch 2.0) make a bit faster    #
# Updated STM 2008-12-24 (Beta Patch 3.0) extendflags          #
# Updated STM 2009-05-29 (Beta Patch 4.0) for mcmaster tutorial #
# Updated STM 2009-12-20 (Release 0)      for Version 3.0.0    #
# Revised J0 2010-04-13 (Release 3.0.1) edit viewer call       #
#                                                                 #
# This is a VLA 6cm dataset that was observed in 1999 to set the #
# flux scale for calibration of the VLA. Included in the program #
# were observations of the planets, including Jupiter.          #

```

```

#                                                                 #
# This is D-configuration data, with resolution of around 14"    #
#                                                                 #
# Includes polarization imaging and analysis                      #
#                                                                 #
#####

import time
import os

print "Jupiter 6cm Interactive Tutorial/Demo Script"
print "Version 2009-12-20 (Version 3.0.0 Release 0)"
print ""

#
#=====
#
# This script has some interactive commands: scriptmode = True
# if you are running it and want it to stop during interactive parts.

scriptmode = True

#=====
#
# Set up some useful variables - these will be set during the script
# also, but if you want to restart the script in the middle here
# they are in one place:

# This will prefix all output file names
prefix='jupiter6cm.demo'

# Clean up old files
os.system('rm -rf '+prefix+'*')

# This is the output MS file name
msfile = prefix + '.ms'

#
#=====
# Calibration variables
#
# Use same prefix as rest of script
calprefix = prefix

# spectral windows to process
usespw = ''
usespwlis = ['0','1']

# prior calibration to apply
usegaincurve = True
gainopacity = 0.0

```

```

# reference antenna 11 (11=VLA:N1)
calrefant = '11'

gtable = calprefix + '.gcal'
ftable = calprefix + '.fluxscale'
atable = calprefix + '.accum'

#
#=====
# Polarization calibration setup
#
dopolcal = True

ptable = calprefix + '.pcal'
xtable = calprefix + '.polx'

# Pol leakage calibrator
poldfield = '0137+331'

# Pol angle calibrator
polxfield = '1331+305'
# At Cband the fractional polarization of this source is 0.112 and
# the R-L PhaseDiff = 66deg (EVPA = 33deg)
polxfpol = 0.112
polxrlpd_deg = 66.0
# Dictionary of IPOL in the spw
polxipol = {'0' : 7.462,
            '1' : 7.510}

# Make Stokes lists for setjy
polxiqyv = {}
for spw in ['0','1']:
    ipol = polxipol[spw]
    fpol = polxfpol
    ppol = ipol*fpol
    rlpd = polxrlpd_deg*pi/180.0
    qpol = ppol*cos(rlpd)
    upol = ppol*sin(rlpd)
    polxiqyv[spw] = [ipol,qpol,upol,0.0]

#
# Split output setup
#
srcname = 'JUPITER'
srcsplitms = calprefix + '.' + srcname + '.split.ms'
calname = '0137+331'
calsplitms = calprefix + '.' + calname + '.split.ms'

#
#=====

```

```
#
# Intensity imaging parameters
#
# Same prefix for this imaging demo output
#
imprefix = prefix

# This is D-config VLA 6cm (4.85GHz) obs
# Check the observational status summary
# Primary beam FWHM = 45'/f_GHz = 557"
# Synthesized beam FWHM = 14"
# RMS in 10min (600s) = 0.06 mJy (thats now, but close enough)

# Set the output image size and cell size (arcsec)
# 4" will give 3.5x oversampling
clncell = [4.,4.]

# 280 pix will cover to 2xPrimaryBeam
# clean will say to use 288 (a composite integer) for efficiency
clnalg = 'clark'
clnmode = ''
# For Cotton-Schwab use
clnmode = 'csclean'
clnimsize = [288,288]

# iterations
clniter = 10000

# Also set flux residual threshold (0.04 mJy)
# From our listobs:
# Total integration time = 85133.2 seconds
# With rms of 0.06 mJy in 600s ==> rms = 0.005 mJy
# Set to 10x thermal rms
clnthreshold=0.05

#
# Filenames
#
imname1 = imprefix + '.clean1'
clnimage1 = imname1+'.image'
clnmodel1 = imname1+'.model'
clnresid1 = imname1+'.residual'
clnmask1 = imname1+'.clean_interactive.mask'

imname2 = imprefix + '.clean2'
clnimage2 = imname2+'.image'
clnmodel2 = imname2+'.model'
clnresid2 = imname2+'.residual'
clnmask2 = imname2+'.clean_interactive.mask'

imname3 = imprefix + '.clean3'
```

```

clnimage3 = imname3+'.image'
clnmodel3 = imname3+'.model'
clnresid3 = imname3+'.residual'
clnmask3 = imname3+'.clean_interactive.mask'

#
# Selfcal parameters
#
# reference antenna 11 (11=VLA:N1)
calrefant = '11'

#
# Filenames
#
selfcaltab1 = imprefix + '.selfcal1.gtable'

selfcaltab2 = imprefix + '.selfcal2.gtable'
smoothcaltab2 = imprefix + '.smoothcal2.gtable'

#
#=====
#
# Polarization imaging parameters
#
# New prefix for polarization imaging output
#
polprefix = prefix + '.polimg'

# Set up clean slightly differently
polclnalg = 'hogbom'
polclnmode = 'csclean'

polimname = polprefix + '.clean'
polimage = polimname+'.image'
polmodel = polimname+'.model'
polresid = polimname+'.residual'
polmask = polimname+'.clean_interactive.mask'

#
# Other files
#
ipolimage = polimage+'.I'
qpolimage = polimage+'.Q'
upolimage = polimage+'.U'

poliimage = polimage+'.poli'
polaimage = polimage+'.pola'

#
#=====
#=====

```



```

# Start processing
#=====
#
# Get to path to the CASA home and strip off the name
pathname=os.environ.get('CASAPATH').split()[0]

# This is where the UVFITS data should be
#fitsdata=pathname+'/data/demo/jupiter6cm.fits'
# Or
#fitsdata=pathname+'/data/nrao/VLA/planets_6cm.fits'
#fitsdata='/home/ballista/casa/devel/data/nrao/VLA/planets_6cm.fits'
#
# Can also be found online at
#http://casa.nrao.edu/Data/VLA/Planets6cm/planets_6cm.fits

# Use version in current directory
fitsdata='planets_6cm.fits'

#
#=====
# Data Import and List
#=====
#
# Import the data from FITS to MS
#
print '--Import--'

# Safest to start from task defaults
default('importuvfits')

print "Use importuvfits to read UVFITS and make an MS"

# Set up the MS filename and save as new global variable
msfile = prefix + '.ms'

print "MS will be called "+msfile

# Use task importuvfits
fitsfile = fitsdata
vis = msfile
importuvfits()

#=====
#
# List a summary of the MS
#
print '--Listobs--'

# Don't default this one and make use of the previous setting of
# vis. Remember, the variables are GLOBAL!

```

```

print "Use listobs to print verbose summary to logger"

# You may wish to see more detailed information, in this case
# use the verbose = True option
verbose = True

listobs()

# You should get in your logger window and in the casapy.log file
# something like:
#
#   Observer: FLUX99      Project:
# Observation: VLA
#
# Data records: 2021424      Total integration time = 85133.2 seconds
#   Observed from  23:15:27   to   22:54:20
#
#   ObservationID = 0      ArrayID = 0
#   Date           Timerange           Scan  FldId  FieldName      SpwIds
# 15-Apr-1999/23:15:26.7 - 23:16:10.0    1     0  0137+331      [0, 1]
#                   23:38:40.0 - 23:48:00.0    2     1  0813+482      [0, 1]
#                   23:53:40.0 - 23:55:20.0    3     2  0542+498      [0, 1]
# 16-Apr-1999/00:22:10.1 - 00:23:49.9    4     3  0437+296      [0, 1]
#                   00:28:23.3 - 00:30:00.1    5     4  VENUS         [0, 1]
#                   00:48:40.0 - 00:50:20.0    6     1  0813+482      [0, 1]
#                   00:56:13.4 - 00:57:49.9    7     2  0542+498      [0, 1]
#                   01:10:20.1 - 01:11:59.9    8     5  0521+166      [0, 1]
#                   01:23:29.9 - 01:25:00.1    9     3  0437+296      [0, 1]
#                   01:29:33.3 - 01:31:10.0   10     4  VENUS         [0, 1]
#                   01:49:50.0 - 01:51:30.0   11     6  1411+522      [0, 1]
#                   02:03:00.0 - 02:04:30.0   12     7  1331+305      [0, 1]
#                   02:17:30.0 - 02:19:10.0   13     1  0813+482      [0, 1]
#                   02:24:20.0 - 02:26:00.0   14     2  0542+498      [0, 1]
#                   02:37:49.9 - 02:39:30.0   15     5  0521+166      [0, 1]
#                   02:50:50.1 - 02:52:20.1   16     3  0437+296      [0, 1]
#                   02:59:20.0 - 03:01:00.0   17     6  1411+522      [0, 1]
#                   03:12:30.0 - 03:14:10.0   18     7  1331+305      [0, 1]
#                   03:27:53.3 - 03:29:39.9   19     1  0813+482      [0, 1]
#                   03:35:00.0 - 03:36:40.0   20     2  0542+498      [0, 1]
#                   03:49:50.0 - 03:51:30.1   21     6  1411+522      [0, 1]
#                   04:03:10.0 - 04:04:50.0   22     7  1331+305      [0, 1]
#                   04:18:49.9 - 04:20:40.0   23     1  0813+482      [0, 1]
#                   04:25:56.6 - 04:27:39.9   24     2  0542+498      [0, 1]
#                   04:42:49.9 - 04:44:40.0   25     8  MARS          [0, 1]
#                   04:56:50.0 - 04:58:30.1   26     6  1411+522      [0, 1]
#                   05:24:03.3 - 05:33:39.9   27     7  1331+305      [0, 1]
#                   05:48:00.0 - 05:49:49.9   28     1  0813+482      [0, 1]
#                   05:58:36.6 - 06:00:30.0   29     8  MARS          [0, 1]
#                   06:13:20.1 - 06:14:59.9   30     6  1411+522      [0, 1]
#                   06:27:40.0 - 06:29:20.0   31     7  1331+305      [0, 1]
#                   06:44:13.4 - 06:46:00.0   32     1  0813+482      [0, 1]

```

#	06:55:06.6 - 06:57:00.0	33	8 MARS	[0, 1]
#	07:10:40.0 - 07:12:20.0	34	6 1411+522	[0, 1]
#	07:28:20.0 - 07:30:10.1	35	7 1331+305	[0, 1]
#	07:42:49.9 - 07:44:30.0	36	8 MARS	[0, 1]
#	07:58:43.3 - 08:00:39.9	37	6 1411+522	[0, 1]
#	08:13:30.0 - 08:15:19.9	38	7 1331+305	[0, 1]
#	08:27:53.4 - 08:29:30.0	39	8 MARS	[0, 1]
#	08:42:59.9 - 08:44:50.0	40	6 1411+522	[0, 1]
#	08:57:09.9 - 08:58:50.0	41	7 1331+305	[0, 1]
#	09:13:03.3 - 09:14:50.1	42	9 NGC7027	[0, 1]
#	09:26:59.9 - 09:28:40.0	43	6 1411+522	[0, 1]
#	09:40:33.4 - 09:42:09.9	44	7 1331+305	[0, 1]
#	09:56:19.9 - 09:58:10.0	45	9 NGC7027	[0, 1]
#	10:12:59.9 - 10:14:50.0	46	8 MARS	[0, 1]
#	10:27:09.9 - 10:28:50.0	47	6 1411+522	[0, 1]
#	10:40:30.0 - 10:42:00.0	48	7 1331+305	[0, 1]
#	10:56:10.0 - 10:57:50.0	49	9 NGC7027	[0, 1]
#	11:28:30.0 - 11:35:30.0	50	10 NEPTUNE	[0, 1]
#	11:48:20.0 - 11:50:10.0	51	6 1411+522	[0, 1]
#	12:01:36.7 - 12:03:10.0	52	7 1331+305	[0, 1]
#	12:35:33.3 - 12:37:40.0	53	11 URANUS	[0, 1]
#	12:46:30.0 - 12:48:10.0	54	10 NEPTUNE	[0, 1]
#	13:00:29.9 - 13:02:10.0	55	6 1411+522	[0, 1]
#	13:15:23.3 - 13:17:10.1	56	9 NGC7027	[0, 1]
#	13:33:43.3 - 13:35:40.0	57	11 URANUS	[0, 1]
#	13:44:30.0 - 13:46:10.0	58	10 NEPTUNE	[0, 1]
#	14:00:46.7 - 14:01:39.9	59	0 0137+331	[0, 1]
#	14:10:40.0 - 14:12:09.9	60	12 JUPITER	[0, 1]
#	14:24:06.6 - 14:25:40.1	61	11 URANUS	[0, 1]
#	14:34:30.0 - 14:36:10.1	62	10 NEPTUNE	[0, 1]
#	14:59:13.4 - 15:00:00.0	63	0 0137+331	[0, 1]
#	15:09:03.3 - 15:10:40.1	64	12 JUPITER	[0, 1]
#	15:24:30.0 - 15:26:20.1	65	9 NGC7027	[0, 1]
#	15:40:10.0 - 15:45:00.0	66	11 URANUS	[0, 1]
#	15:53:50.0 - 15:55:20.0	67	10 NEPTUNE	[0, 1]
#	16:18:53.4 - 16:19:49.9	68	0 0137+331	[0, 1]
#	16:29:10.1 - 16:30:49.9	69	12 JUPITER	[0, 1]
#	16:42:53.4 - 16:44:30.0	70	11 URANUS	[0, 1]
#	16:54:53.4 - 16:56:40.0	71	9 NGC7027	[0, 1]
#	17:23:06.6 - 17:30:40.0	72	2 0542+498	[0, 1]
#	17:41:50.0 - 17:43:20.0	73	3 0437+296	[0, 1]
#	17:55:36.7 - 17:57:39.9	74	4 VENUS	[0, 1]
#	18:19:23.3 - 18:20:09.9	75	0 0137+331	[0, 1]
#	18:30:23.3 - 18:32:00.0	76	12 JUPITER	[0, 1]
#	18:44:49.9 - 18:46:30.0	77	9 NGC7027	[0, 1]
#	18:59:13.3 - 19:00:59.9	78	2 0542+498	[0, 1]
#	19:19:10.0 - 19:21:20.1	79	5 0521+166	[0, 1]
#	19:32:50.1 - 19:34:29.9	80	3 0437+296	[0, 1]
#	19:39:03.3 - 19:40:40.1	81	4 VENUS	[0, 1]
#	20:08:06.7 - 20:08:59.9	82	0 0137+331	[0, 1]
#	20:18:10.0 - 20:19:50.0	83	12 JUPITER	[0, 1]

```

#           20:33:53.3 - 20:35:40.1    84      1 0813+482      [0, 1]
#           20:40:59.9 - 20:42:40.0    85      2 0542+498      [0, 1]
#           21:00:16.6 - 21:02:20.1    86      5 0521+166      [0, 1]
#           21:13:53.4 - 21:15:29.9    87      3 0437+296      [0, 1]
#           21:20:43.4 - 21:22:30.0    88      4 VENUS        [0, 1]
#           21:47:26.7 - 21:48:20.1    89      0 0137+331      [0, 1]
#           21:57:30.0 - 21:59:10.0    90     12 JUPITER      [0, 1]
#           22:12:13.3 - 22:14:00.1    91      2 0542+498      [0, 1]
#           22:28:33.3 - 22:30:19.9    92      4 VENUS        [0, 1]
#           22:53:33.3 - 22:54:19.9    93      0 0137+331      [0, 1]
#
# Fields: 13
#   ID   Name           Right Ascension  Declination  Epoch
#   0    0137+331       01:37:41.30    +33.09.35.13 J2000
#   1    0813+482       08:13:36.05    +48.13.02.26 J2000
#   2    0542+498       05:42:36.14    +49.51.07.23 J2000
#   3    0437+296       04:37:04.17    +29.40.15.14 J2000
#   4    VENUS          04:06:54.11    +22.30.35.91 J2000
#   5    0521+166       05:21:09.89    +16.38.22.05 J2000
#   6    1411+522       14:11:20.65    +52.12.09.14 J2000
#   7    1331+305       13:31:08.29    +30.30.32.96 J2000
#   8    MARS           14:21:41.37    -12.21.49.45 J2000
#   9    NGC7027        21:07:01.59    +42.14.10.19 J2000
#  10    NEPTUNE        20:26:01.14    -18.54.54.21 J2000
#  11    URANUS         21:15:42.83    -16.35.05.59 J2000
#  12    JUPITER        00:55:34.04    +04.45.44.71 J2000
#
# Spectral Windows: (2 unique spectral windows and 1 unique polarization setups)
#   SpwID #Chans Frame Ch1(MHz)   Resoln(kHz) TotBW(kHz) Ref(MHz)   Corrs
#   0           1 TOPO  4885.1     50000      50000     4885.1     RR RL LR LL
#   1           1 TOPO  4835.1     50000      50000     4835.1     RR RL LR LL
#
# Feeds: 28: printing first row only
#   Antenna   Spectral Window   # Receptors   Polarizations
#   1         -1                2              [      R, L]
#
# Antennas: 27:
#   ID   Name   Station   Diam.   Long.   Lat.
#   0    1     VLA:W9   25.0 m  -107.37.25.1 +33.53.51.0
#   1    2     VLA:N9   25.0 m  -107.37.07.8 +33.54.19.0
#   2    3     VLA:N3   25.0 m  -107.37.06.3 +33.54.04.8
#   3    4     VLA:N5   25.0 m  -107.37.06.7 +33.54.08.0
#   4    5     VLA:N2   25.0 m  -107.37.06.2 +33.54.03.5
#   5    6     VLA:E1   25.0 m  -107.37.05.7 +33.53.59.2
#   6    7     VLA:E2   25.0 m  -107.37.04.4 +33.54.01.1
#   7    8     VLA:N8   25.0 m  -107.37.07.5 +33.54.15.8
#   8    9     VLA:E8   25.0 m  -107.36.48.9 +33.53.55.1
#   9   10     VLA:W3   25.0 m  -107.37.08.9 +33.54.00.1
#  10   11     VLA:N1   25.0 m  -107.37.06.0 +33.54.01.8
#  11   12     VLA:E6   25.0 m  -107.36.55.6 +33.53.57.7
#  12   13     VLA:W7   25.0 m  -107.37.18.4 +33.53.54.8

```

```

# 13 14 VLA:E4 25.0 m -107.37.00.8 +33.53.59.7
# 14 15 VLA:N7 25.0 m -107.37.07.2 +33.54.12.9
# 15 16 VLA:W4 25.0 m -107.37.10.8 +33.53.59.1
# 16 17 VLA:W5 25.0 m -107.37.13.0 +33.53.57.8
# 17 18 VLA:N6 25.0 m -107.37.06.9 +33.54.10.3
# 18 19 VLA:E7 25.0 m -107.36.52.4 +33.53.56.5
# 19 20 VLA:E9 25.0 m -107.36.45.1 +33.53.53.6
# 21 22 VLA:W8 25.0 m -107.37.21.6 +33.53.53.0
# 22 23 VLA:W6 25.0 m -107.37.15.6 +33.53.56.4
# 23 24 VLA:W1 25.0 m -107.37.05.9 +33.54.00.5
# 24 25 VLA:W2 25.0 m -107.37.07.4 +33.54.00.9
# 25 26 VLA:E5 25.0 m -107.36.58.4 +33.53.58.8
# 26 27 VLA:N4 25.0 m -107.37.06.5 +33.54.06.1
# 27 28 VLA:E3 25.0 m -107.37.02.8 +33.54.00.5
#

```

```
# Tables:
```

```

# MAIN 2021424 rows
# ANTENNA 28 rows
# DATA_DESCRIPTION 2 rows
# DOPPLER <absent>
# FEED 28 rows
# FIELD 13 rows
# FLAG_CMD <empty>
# FREQ_OFFSET <absent>
# HISTORY 7058 rows
# OBSERVATION 1 row
# POINTING 2604 rows
# POLARIZATION 1 row
# PROCESSOR <empty>
# SOURCE <empty> (see FIELD)
# SPECTRAL_WINDOW 2 rows
# STATE <empty>
# SYSCAL <absent>
# WEATHER <absent>
#

```

```
#
```

```
#=====
```

```
# Data Examination and Flagging
```

```
#=====
```

```
#
```

```
# Use Plotxy to interactively flag the data
```

```
#
```

```
print '--Plotxy--'
```

```
default('plotxy')
```

```
print "Now we use plotxy to examine and interactively flag data"
```

```
vis = msfile
```

```
# The fields we are interested in: 1331+305,JUPITER,0137+331
```

```
selectdata = True
```

```

# First we do the primary calibrator
field = '1331+305'

# Plot only the RR and LL for now
correlation = 'RR LL'

# As of 2.3.0 (Patch 3) you can extend the flags to the cross-correlations
# But this slows things down immensely
#extendflag = T
#extendcorr = 'all'

# Plot amplitude vs. uvdist
xaxis = 'uvdist'
yaxis = 'amp'
multicolor = 'both'

# Use the field name as the title
selectplot = True
title = field+" "

iteration = ''

plotxy()

print ""
print "-----"
print "Plotxy"
print "Showing 1331+305 RR LL for all antennas"
print "Use MarkRegion then draw boxes around points to flag"
print "You can use ESC to drop last drawn box"
print "When happy with boxes, hit Flag to flag"
print "You can repeat as necessary"
print ""
#print "NOTE: These flags will extend to the RL LR cross-hands"
#print "Because of this the flagging will be slower than otherwise"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

# You can also use flagdata to do this non-interactively
# (see below)

# Now look at the cross-polar products
correlation = 'RL LR'
extendflag = F

plotxy()

print ""

```

```

print "-----"
print "Looking at RL LR"
print "Now flag any remaining bad data here"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

#-----
# Now do calibrator 0137+331
field = '0137+331'
correlation = 'RR LL'
xaxis = 'uvdist'
spw = ''
iteration = ''
antenna = ''

# As of 2.3.0 (Patch 3) you can extend the flags to the cross-correlations
# But this slows things down immensely
#extendflag = T
#extendcorr = 'all'

title = field+" "

plotxy()

# You'll see a bunch of bad data along the bottom near zero amp
# Draw a box around some of it and use Locate
# Looks like much of it is Antenna 9 (ID=8) in spw=1

print ""
print "-----"
print "Plotting 0137+331 RR LL all antennas"
print "You see bad data along bottom"
print "Mark a box around a bit of it and hit Locate"
print "Look in logger to see what it is"
print "You see much is Antenna 9 (ID=8) in spw 1"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

xaxis = 'time'
spw = '1'
correlation = ''
extendflag = F

# Note that the strings like antenna='9' first try to match the
# NAME which we see in listobs was the number '9' for ID=8.
# So be careful here (why naming antennas as numbers is bad).
antenna = '9'

```

```

plotxy()

# YES! the last 4 scans are bad.  Box 'em and flag.

print ""
print "-----"
print "Plotting vs. time antenna='9' and spw='1' "
print "Box up last 4 scans which are bad and Flag"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

# Go back and clean up
xaxis = 'uvdist'
spw = ''
antenna = ''

correlation = 'RR LL'

# Note that RL,LR are too weak to clip on.
# As of 2.3.0 (Patch 3) you can extend the flags to the cross-correlations
# But this slows things down immensely
#extendflag = T
#extendcorr = 'all'

plotxy()

# Box up the bad low points (basically a clip below 0.52) and flag

print ""
print "-----"
print "Back to all data"
print "Clean up remaining bad points"
print ""
#print "NOTE: These flags will extend to the RL LR cross-hands"
#print "Because of this the flagging will be slower than otherwise"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

#-----
# Finally, do JUPITER
field = 'JUPITER'
correlation = 'RR LL'
iteration = ''
xaxis = 'uvdist'

title = field+" "

```



```

plotxy()

# Here you will see that the final scan at 22:00:00 UT is bad
# Draw a box around it and flag it!

print ""
print "-----"
print "Now plot JUPITER versus uvdist"
print "Lots of bad stuff near bottom"
print "Lets go and find it - try Locate"
print "Looks like lots of different antennas but at same time"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

correlation = ''
extendflag = F
xaxis = 'time'

plotxy()

# Here you will see that the final scan at 22:00:00 UT is bad
# Draw a box around it and flag it!

print ""
print "-----"
print "Now plotting vs. time"
print "See bad scan at end - flag it!"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

# Now look at whats left
correlation = 'RR LL'
# As of 2.3.0 (Patch 3) you can extend the flags to the cross-correlations
# But this slows things down immensely
#extendflag = T
#extendcorr = 'all'

xaxis = 'uvdist'
spw = '1'
antenna = ''
iteration = 'antenna'

plotxy()

# As you step through, you will see that Antenna 9 (ID=8) is often
# bad in this spw. If you box and do Locate (or remember from

```

```

# 0137+331) its probably a bad time.

print ""
print "-----"
print "Looking now at SPW 1"
print "Now we set iteration to Antenna"
print "Step through antennas with Next"
print "See bad Antenna 9 (ID 8) as in 0137+331"
print "Do not flag yet, we will isolate this next"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

# The easiset way to kill it:

antenna = '9'
iteration = ''
xaxis = 'time'
correlation = ''
extendflag = F

plotxy()

# Draw a box around all points in the last bad scans and flag 'em!

print ""
print "-----"
print "Now plotting vs. time antenna 9 spw 1"
print "Box up the bad scans and Flag"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

# Now clean up the rest
xaxis = 'uvdist'
correlation = 'RR LL'
# As of 2.3.0 (Patch 3) you can extend the flags to the cross-correlations
# But this slows things down immensely
#extendflag = T
#extendcorr = 'all'

antenna = ''
spw = ''

# You will be drawing many tiny boxes, so remember you can
# use the ESC key to get rid of the most recent box if you
# make a mistake.

plotxy()

```

```

# Note that the end result is we've flagged lots of points
# in RR and LL. We will rely upon imager to ignore the
# RL LR for points with RR LL flagged!

print ""
print "-----"
print "Final cleanup of JUPITER data"
print "Back to uvdist plot, see remaining bad data"
print "You can draw little boxes around the outliers and Flag"
print "Depends how patient you are in drawing boxes!"
print "Could also use Locate to find where they come from"
print ""
#print "NOTE: These flags will extend to the RL LR cross-hands"
#print "Because of this the flagging will be slower than otherwise"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

print "Done with plotxy!"

#
#=====
#
# Use Flagmanager to save a copy of the flags so far
#
print '--Flagmanager--'
default('flagmanager')

print "Now will use flagmanager to save a copy of the flags we just made"
print "These are named xyflags"

vis = msfile
mode = 'save'
versionname = 'xyflags'
comment = 'Plotxy flags'
merge = 'replace'

flagmanager()

#=====
#
# Use Flagmanager to list all saved versions
#
print '--Flagmanager--'
default('flagmanager')

print "Now will use flagmanager to list all the versions we saved"

vis = msfile

```

```

mode = 'list'

flagmanager()

#
# Done Flagging
print '--Done with flagging--'

#
#=====
# Calibration
#=====
#
# Set the fluxes of the primary calibrator(s)
#
print '--Setjy--'
default('setjy')

print "Use setjy to set flux of 1331+305 (3C286)"

vis = msfile

#
# 1331+305 = 3C286 is our primary calibrator
field = '1331+305'

# Setjy knows about this source so we dont need anything more

setjy()

#
# You should see something like this in the logger and casapy.log file:
#
# 1331+305 spwid= 0 [I=7.462, Q=0, U=0, V=0] Jy, (Perley-Taylor 99)
# 1331+305 spwid= 1 [I=7.51, Q=0, U=0, V=0] Jy, (Perley-Taylor 99)
#

print "Look in logger for the fluxes (should be 7.462 and 7.510 Jy)"

#
#=====
#
# Initial gain calibration
#
print '--Gaincal--'
default('gaincal')

print "Solve for antenna gains on 1331+305 and 0137+331"
print "We have 2 single-channel continuum spw"
print "Do not want bandpass calibration"

```

```

vis = msfile

# set the name for the output gain caltable
caltable = gtable

print "Output gain cal table will be "+gtable

# Gain calibrators are 1331+305 and 0137+331 (FIELD_ID 7 and 0)
# We have 2 IFs (SPW 0,1) with one channel each

# selection is via the field and spw strings
field = '1331+305,0137+331'
spw = ''

# a-priori calibration application
gaincurve = usegaincurve
opacity = gainopacity

# scan-based G solutions for both amplitude and phase
gaintype = 'G'
calmode = 'ap'

# one solution per scan
solint = 'inf'
combine = ''

# do not apply parallactic angle correction (yet)
parang = False

# reference antenna
refant = calrefant

# minimum SNR 3
minsnr = 3

gaincal()

#
#=====
#
# Bootstrap flux scale
#
print '--Fluxscale--'
default('fluxscale')

print "Use fluxscale to rescale gain table to make new one"

vis = msfile

# set the name for the output rescaled caltable
fluxtable = ftable

```

```

print "Output scaled gain cal table is "+ftable

# point to our first gain cal table
caltable = gtable

# we will be using 1331+305 (the source we did setjy on) as
# our flux standard reference
reference = '1331+305'

# we want to transfer the flux to our other gain cal source 0137+331
# to bring its gain amplitues in line with the absolute scale
transfer = '0137+331'

fluxscale()

# You should see in the logger something like:
#Flux density for 0137+331 in SpW=0 is:
# 5.42575 +/- 0.00285011 (SNR = 1903.7, nAnt= 27)
#Flux density for 0137+331 in SpW=1 is:
# 5.46569 +/- 0.00301326 (SNR = 1813.88, nAnt= 27)

#
#-----
# Plot calibration
#
print '--PlotCal--'
default('plotcal')

showgui = True

caltable = ftable
multiplot = True
yaxis = 'amp'

showgui = True

plotcal()

print ""
print "-----"
print "Plotcal"
print "Looking at amplitude in cal-table "+caltable

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

#
# Now go back and plot to file
#

```

```

showgui = False

yaxis = 'amp'

#figfile = caltable + '.plotcal.amp.png'
#print "Plotting calibration to file "+figfile
#saveinputs('plotcal',caltable.plotcal.amp.saved')
#plotcal()

yaxis = 'phase'

#figfile = caltable + '.plotcal.phase.png'
#print "Plotting calibration to file "+figfile
#saveinputs('plotcal',caltable.plotcal.phase.saved')
#plotcal()

#
#=====
# Polarization Calibration
#=====
#
if (dopolcal):
    print '--Polcal (D)--'
    default('polcal')

    print "Solve for polarization leakage on 0137+331"
    print "Pretend it has unknown polarization"

    vis = msfile

    # Start with the un-fluxscaled gain table
    gaintable = gtable

    # use settings from gaincal
    gaincurve = usegaincurve
    opacity = gainopacity

    # Output table
    caltable = ptable

    # Use a 3C48 tracked through a range of PA
    field = '0137+331'
    spw = ''

    # No need for further selection
    selectdata=False

    # Polcal mode (D+QU = unknown pol for D)
    poltype = 'D+QU'

    # One solution for entire dataset

```

```

solint = 'inf'
combine = 'scan'

# reference antenna
refant = calrefant

# minimum SNR 3
minsnr = 3

#saveinputs('polcal',calprefix+'.polcal.saved')
polcal()

#=====
#
# List polcal solutions
#
print '--Listcal (PolD)--'

listfile = caltable + '.list'

print "Listing calibration to file "+listfile

listcal()

#=====
#
# Plot polcal solutions
#
print '--Plotcal (PolD)--'

iteration = ''
showgui = False

xaxis = 'antenna'
yaxis = 'amp'

showgui = True
figfile = ''

plotcal()

print "These are the amplitudes of D-terms versus antenna"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

# Now plot to files
showgui = False

#figfile = caltable + '.plotcal.antamp.png'

```



```

#print "Plotting calibration to file "+figfile
#saveinputs('plotcal',caltable+'.plotcal.antamp.saved')
#plotcal()

xaxis = 'antenna'
yaxis = 'phase'

#figfile = caltable + '.plotcal.antphase.png'
#print "Plotting calibration to file "+figfile
#saveinputs('plotcal',caltable+'.plotcal.antphase.saved')
#plotcal()

xaxis = 'antenna'
yaxis = 'snr'

#figfile = caltable + '.plotcal.antsnr.png'
#print "Plotting calibration to file "+figfile
#saveinputs('plotcal',caltable+'.plotcal.antsnr.saved')
#plotcal()

xaxis = 'real'
yaxis = 'imag'

#figfile = caltable + '.plotcal.reim.png'
#print "Plotting calibration to file "+figfile
#saveinputs('plotcal',caltable+'.plotcal.reim.saved')
#plotcal()

#=====
# Do Chi (X) pol angle calibration
#=====
# First set the model
print '--Setjy--'
default('setjy')

vis = msfile

print "Use setjy to set IQU fluxes of "+polxfield
field = polxfield

for spw in usespwlist:
    fluxdensity = polxiquv[spw]

    #saveinputs('setjy',calprefix+'.setjy.polspw.'+spw+'.saved')
    setjy()

#
# Polarization (X-term) calibration
#
print '--PolCal (X)--'
default('polcal')

```

```

print "Polarization R-L Phase Calibration (linear approx)"

vis = msfile

# Start with the G and D tables
gaintable = [gtable,ptable]

# use settings from gaincal
gaincurve = usegaincurve
opacity = gainopacity

# Output table
caltable = xtable

# previously set with setjy
field = polxfield
spw = ''

selectdata=False

# Solve for Chi
poltype = 'X'
solint = 'inf'
combine = 'scan'

# reference antenna
refant = calrefant

# minimum SNR 3
minsnr = 3

#saveinputs('polcal',calprefix+'.polcal.X.saved')
polcal()

#=====
# Apply the Calibration
#=====
#
# Interpolate the gains onto Jupiter (and others)
#
# print '--Accum--'
# default('accum')
#
# print "This will interpolate the gains onto Jupiter"
#
# vis = msfile
#
# tablein = ''
# incrtable = ftable
# calfield = '1331+305, 0137+331'

```

```

#
# # set the name for the output interpolated caltable
# caltable = atable
#
# print "Output cumulative gain table will be "+atable
#
# # linear interpolation
# interp = 'linear'
#
# # make 10s entries
# accumtime = 10.0
#
# accum()
#
# NOTE: bypassing this during testing
atable = ftable

# #=====
#
# Correct the data
# (This will put calibrated data into the CORRECTED_DATA column)
#
print '--ApplyCal--'
default('applycal')

print "This will apply the calibration to the DATA"
print "Fills CORRECTED_DATA"

vis = msfile

# Start with the interpolated fluxscale/gain table
gaintable = [atable,ptable,xtable]

# use settings from gaincal
gaincurve = usegaincurve
opacity = gainopacity

# select the fields
field = '1331+305,0137+331,JUPITER'
spw = ''
selectdata = False

# IMPORTANT set parang=True for polarization
parang = True

# do not need to select subset since we did accum
# (note that correct only does 'nearest' interp)
gainfield = ''

applycal()

```

```

#
#=====
#
# Now split the Jupiter target data
#
print '--Split Jupiter--'
default('split')

vis = msfile

# Now we write out the corrected data to a new MS

# Select the Jupiter field
field = srcname
spw = ''

# pick off the CORRECTED_DATA column
datacolumn = 'corrected'

# Make an output vis file
outputvis = srcsplitms

print "Split "+field+" data into new ms "+srcsplitms

split()

# Also split out 0137+331 as a check
field = calname

outputvis = calsplitms

print "Split "+field+" data into new ms "+calsplitms

split()

#=====
# Force scratch column creation so plotxy will work
#
vis = srcsplitms
clearcal()

vis = calsplitms
clearcal()

#=====
# Use Plotxy to look at the split calibrated data
#
print '--Plotxy--'
default('plotxy')

selectdata = True

```

```

correlation = 'RR LL'
xaxis = 'uvdist'
datacolumn = 'data'
multicolor = 'both'
iteration = ''
selectplot = True
field = 'JUPITER'

#vis = srcsplitms
#interactive = True
#yaxis = 'amp'
#title = field+" "
#
# Plotxy interactively if desired
#plotxy()
#
#print ""
#print "-----"
#print "Plotting JUPITER corrected visibilities"
#print "Look for outliers"

# Pause script if you are running in scriptmode
#if scriptmode:
#   user_check=raw_input('Return to continue script\n')
#
# Now go back and plot to files
interactive = False

#
# First the target
#
vis = srcsplitms
field = srcname
yaxis = 'amp'
# Use the field name as the title
title = field+" "

#figfile = vis + '.plotxy.amp.png'
#print "Plotting to file "+figfile
#saveinputs('plotxy',vis+'.plotxy.amp.saved')
#plotxy()

yaxis = 'phase'
# Use the field name as the title

#figfile = vis + '.plotxy.phase.png'
#print "Plotting to file "+figfile
#saveinputs('plotxy',vis+'.plotxy.phase.saved')
#plotxy()

#

```

```

# Now the calibrator
#
vis = calsplitms
field = calname
yaxis = 'amp'
# Use the field name as the title
title = field+" "

#figfile = vis + '.plotxy.amp.png'
#print "Plotting to file "+figfile
#saveinputs('plotxy',vis+'.plotxy.amp.saved')
#plotxy()

yaxis = 'phase'

#figfile = vis + '.plotxy.phase.png'
#print "Plotting to file "+figfile
#saveinputs('plotxy',vis+'.plotxy.phase.saved')
#plotxy()

print 'Calibration completed'
#
#=====
#
# Intensity Imaging/Selfcal
#
#=====
#
# Make the scratch columns in the split ms
#
print '--Clearcal--'
default('clearcal')

vis = srcsplitms

clearcal()

print "Created scratch columns for MS "+vis
print ""
#
#=====
# FIRST CLEAN / SELFCAL CYCLE
#=====
#
# Now clean an image of Jupiter
# NOTE: this uses the new combined invert/clean/mosaic task Patch 2
#
print '--Clean 1--'
default('clean')

# Pick up our split source data

```

```
vis = srcsplitms

# Make an image root file name
imagenname = imname1

print "Output images will be prefixed with "+imname1

# Set up the output continuum image (single plane mfs)
mode = 'mfs'
stokes = 'I'

print "Will be a single MFS continuum image"

# NOTE: current version field='' doesnt work
field = '*'

# Combine all spw
spw = ''

# Imaging mode params
psfmode = clnalg
imagermode = clnmode

# Imsize and cell
imsize = clnimsize
cell = clncell

# NOTE: will eventually have an imadvise task to give you this
# information

# Standard gain factor 0.1
gain = 0.1

# Fix maximum number of iterations and threshold
niter = clniter
threshold = clnthreshold

# Note - we can change niter and threshold interactively
# during clean

# Set up the weighting
# Use Briggs weighting (a moderate value, on the uniform side)
weighting = 'briggs'
robust = 0.5

# No clean mask or box
mask = ''

# Use interactive clean mode
interactive = True
```

```

# We will be doing selfcal
calready = True

# Moderate number of iter per interactive cycle
npercycle = 100

saveinputs('clean',imagename+'.clean.saved')
clean()

# When the interactive clean window comes up, use the right-mouse
# to draw rectangles around obvious emission double-right-clicking
# inside them to add to the flag region. You can also assign the
# right-mouse to polygon region drawing by right-clicking on the
# polygon drawing icon in the toolbar. When you are happy with
# the region, click 'Done Flagging' and it will go and clean another
# 100 iterations. When done, click 'Stop'.

print ""
print "-----"
print "Clean"
print "Final clean model is "+clnmodel1
print "Final restored clean image is "+clnimage1
print "The clean residual image is "+clnresid1
print "Your final clean mask is "+clnmask1

print ""
print "This is the final restored clean image in the viewer"
print "Zoom in and set levels to see faint emission"
print "Use rectangle drawing tool to box off source"
print "Double-click inside to print statistics"
print "Move box on-source and get the max"
print "Calcualte DynRange = MAXon/RMSoff"
print "I got 1.060/0.004 = 270"
print "Still not as good as it can be - lets selfcal"
print "Close viewer panel when done"

#
#-----
#
# If you did not do interactive clean, bring up viewer manually
viewer(clnimage1)

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

# You can use the right-mouse to draw a box in the lower right
# corner of the image away from emission, the double-click inside
# to bring up statistics. Use the right-mouse to grab this box
# and move it up over Jupiter and double-click again. You should
# see stuff like this in the terminal:

```



```

#
# jupiter6cm.demo.clean1.image      (Jy/beam)
#
# n          Std Dev      RMS          Mean          Variance      Sum
# 4712       0.003914     0.003927   0.0003205    1.532e-05    1.510
#
# Flux       Med |Dev|     IntQtlRng   Median         Min           Max
# 0.09417    0.002646     0.005294   0.0001885    -0.01125     0.01503
#
#
# On Jupiter:
#
# n          Std Dev      RMS          Mean          Variance      Sum
# 3640       0.1007        0.1027     0.02023      0.01015      73.63
#
# Flux       Med |Dev|     IntQtlRng   Median         Min           Max
# 4.592      0.003239     0.007120   0.0001329    -0.01396     1.060
#
# Estimated dynamic range = 1.060 / 0.003927 = 270 (poor)
#
# Note that the exact numbers you get will depend on how deep you
# take the interactive clean and how you draw the box for the stats.

#=====
#
# Do some non-interactive image statistics
print '--Imstat--'
default('imstat')

imagenname = clnimage1
on_statistics1 = imstat()

# Now do stats in the lower right corner of the image
# remember clnimsize = [288,288]
box = '216,1,287,72'
off_statistics1 = imstat()

# Pull the max and rms from the clean image
thistest_immax=on_statistics1['max'][0]
print ' Found : Max in image = ',thistest_immax
thistest_imrms=off_statistics1['rms'][0]
print ' Found : rms in image = ',thistest_imrms
print ' Clean image Dynamic Range = ',thistest_immax/thistest_imrms
print ''
#
#-----
#
# Self-cal using clean model
#
# Note: clean will have left FT of model in the MODEL_DATA column
# If you've done something in between, can use the ft task to

```

```

# do this manually.
#
print '--SelfCal 1--'
default('gaincal')

vis = srcsplitms

print "Will self-cal using MODEL_DATA left in MS by clean"

# New gain table
caltable = selfcaltab1

print "Will write gain table "+selfcaltab1

# Don't need a-priori calcs
selectdata = False
gaincurve = False
opacity = 0.0

# This choice seemed to work
refant = calrefant

# Do amp and phase
gaintype = 'G'
calmode = 'ap'

# Do 30s solutions with SNR>1
solint = 30.0
minsnr = 1.0
print "Calibrating amplitudes and phases on 30s timescale"

# Do not need to normalize (let gains float)
solnorm = False

gaincal()

#
#-----
# It is useful to put this up in plotcal
#
#
print '--PlotCal--'
default('plotcal')

caltable = selfcaltab1
multiplot = True
yaxis = 'amp'

plotcal()

print ""

```

```

print "-----"
print "Plotcal"
print "Looking at amplitude in self-cal table "+caltable

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

yaxis = 'phase'

plotcal()

print ""
print "-----"
print "Plotcal"
print "Looking at phases in self-cal table "+caltable

#
# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

#
#-----
#
# Correct the data (no need for interpolation this stage)
#
print '--ApplyCal--'
default('applycal')

vis = srcsplitms

print "Will apply self-cal table to over-write CORRECTED_DATA in MS"

gaintable = selfcaltab1

gaincurve = False
opacity = 0.0
field = ''
spw = ''
selectdata = False

calwt = True

applycal()

# Self-cal is now in CORRECTED_DATA column of split ms
#=====
# Use Plotxy to look at the self-calibrated data
#
#print '--Plotxy--'

```

```

#default('plotxy')
#
#vis = srcsplitms
#selectdata = True
#field = 'JUPITER'
#correlation = 'RR LL'
#xaxis = 'uvdist'
#yaxis = 'amp'
#datacolumn = 'corrected'
#multicolor = 'both'
#selectplot = True
#title = field+" "
#
#iteration = ''
#
#plotxy()
#
#print ""
#print "-----"
#print "Plotting JUPITER self-corrected visibilities"
#print "Look for outliers, and you can flag them"

# Pause script if you are running in scriptmode
#if scriptmode:
#   user_check=raw_input('Return to continue script\n')
#

#
#=====
# SECOND CLEAN / SELFCAL CYCLE
#=====
#
print '--Clean 2--'
default('clean')

print "Now clean on self-calibrated data"

vis = srcsplitms

imagenname = imname2

field = '*'
spw = ''
mode = 'mfs'
gain = 0.1

# Imaging mode params
psfmode = clnalg
imagermode = clnmode
imsize = clnimsize
cell = clncell

```

```

niter = clniter
threshold = clnthreshold

weighting = 'briggs'
robust = 0.5

mask = ''
interactive = True
npercycle = 100
calready = True

saveinputs('clean',imagename+'.clean.saved')
clean()

print ""
print "-----"
print "Clean"
print "Final clean model is "+clnmodel2
print "Final restored clean image is "+clnimage2
print "The clean residual image is "+clnresid2
print "Your final clean mask is "+clnmask2

print ""
print "This is the final restored clean image in the viewer"
print "Zoom in and set levels to see faint emission"
print "Use rectangle drawing tool to box off source"
print "Double-click inside to print statistics"
print "Move box on-source and get the max"
print "Calculate DynRange = MAXon/RMSoff"
print "This time I got 1.050 / 0.001 = 1050 (better)"
print "Still not as good as it can be - you can try selfcal again"
print "We will stop here"
print "Close viewer panel when done"

#
#-----
#
# If you did not do interactive clean, bring up viewer manually
viewer(clnimage2)

# Pause script if you are running in scriptmode
if scriptmode:
    user_check=raw_input('Return to continue script\n')

# jupiter6cm.demo.clean2.image      (Jy/beam)
#
# n          Std Dev      RMS          Mean          Variance      Sum
# 5236       0.001389      0.001390     3.244e-05     1.930e-06     0.1699
#
# Flux       Med |Dev|      IntQtlRng    Median         Min           Max
# 0.01060    0.0009064    0.001823    -1.884e-05    -0.004015     0.004892

```

```

#
#
# On Jupiter:
#
# n          Std Dev      RMS          Mean          Variance      Sum
# 5304        0.08512      0.08629      0.01418      0.007245     75.21
#
# Flux        Med |Dev|      IntQtlRng     Median         Min           Max
# 4.695        0.0008142     0.001657     0.0001557     -0.004526    1.076
#
# Estimated dynamic range = 1.076 / 0.001389 = 775 (better)
#
# Note that the exact numbers you get will depend on how deep you
# take the interactive clean and how you draw the box for the stats.
#
print ""
print "-----"
print "After this script is done you can continue on with"
print "more self-cal, or try different cleaning options"

#
#=====
# Image Analysis
#=====
#
# Can do some image statistics if you wish
print '--Imstat (Cycle 2)--'
default('imstat')

imagename = clnimage2
on_statistics2 = imstat()

# Now do stats in the lower right corner of the image
# remember clnimsize = [288,288]
box = '216,1,287,72'
off_statistics2 = imstat()

# Pull the max and rms from the clean image
thistest_immax=on_statistics2['max'][0]
print ' Found : Max in image = ',thistest_immax
thistest_imrms=off_statistics2['rms'][0]
print ' Found : rms in image = ',thistest_imrms
print ' Clean image Dynamic Range = ',thistest_immax/thistest_imrms
print ''

#=====
#
# Print results and regression versus previous runs
#
print ""
print ' Final Jupiter results '

```

```

print ' ===== '
print ''
# Pull the max and rms from the clean image
thistest_immax=on_statistics2['max'][0]
oldtest_immax = 1.07732224464
print ' Clean image ON-SRC max = ',thistest_immax
print ' Previously found to be = ',oldtest_immax
diff_immax = abs((oldtest_immax-thistest_immax)/oldtest_immax)
print ' Difference (fractional) = ',diff_immax

print ''
thistest_imrms=off_statistics2['rms'][0]
oldtest_imrms = 0.0010449
print ' Clean image OFF-SRC rms = ',thistest_imrms
print ' Previously found to be = ',oldtest_imrms
diff_imrms = abs((oldtest_imrms-thistest_imrms)/oldtest_imrms)
print ' Difference (fractional) = ',diff_imrms

print ''
print ' Final Clean image Dynamic Range = ',thistest_immax/thistest_imrms
print ''
print '--- Done with I Imaging and Selfcal---'

#
#=====
# Polarization Imaging
#=====
#
print '--Clean (Polarization)--'
default('clean')

print "Now clean polarized data"

vis = srcsplitms

imagenname = polimname

field = '*'
spw = ''
mode = 'mfs'
gain = 0.1

# Polarization
stokes = 'IQUV'

psfmode = polclnalg
imagermode = polclnmode

niter = clniter
threshold = clnthreshold

```

```

imsize = clnimage
cell = clncell

weighting = 'briggs'
robust = 0.5

interactive = True
npercycle = 100

saveinputs('clean',imagename+'.clean.saved')
clean()

print ""
print "-----"
print "Clean"
print "Final restored clean image is "+polimage
print "Final clean model is "+polmodel
print "The clean residual image is "+polresid
print "Your final clean mask is "+polmask

#
#=====
# Image Analysis
#=====
#
# Polarization statistics
print '--Final Pol Imstat--'
default('imstat')

imagename = polimage

on_statistics = {}
off_statistics = {}

# lower right corner of the image (clnimage = [288,288])
onbox = ''
# lower right corner of the image (clnimage = [288,288])
offbox = '216,1,287,72'

for stokes in ['I','Q','U','V']:
    box = onbox
    on_statistics[stokes] = imstat()
    box = offbox
    off_statistics[stokes] = imstat()

#
# Peel off some Q and U planes
#
print '--Immath--'
default('immath')

```



```

mode = 'evalexpr'

stokes = 'I'
outfile = ipolimage
imagenname = polimage
expr = 'IMO'

immath()
print "Created I image "+outfile

stokes = 'Q'
outfile = qpolimage

immath()
print "Created Q image "+outfile

stokes = 'U'
outfile = upolimage

immath()
print "Created U image "+outfile

#
#-----
# Now make POLI and POLA images
#
# Use our rms above for debiasing
mysigma = 0.5*( off_statistics['Q']['rms'][0] + off_statistics['U']['rms'][0] )
#sigma = str(mysigma)+'Jy/beam'
# This does not work well yet
sigma = '0.0Jy/beam'

#stokes = ''
#imagenname = [qpolimage,upolimage]
#outfile = poliimage
#mode = 'poli'
#immath()
#
#outfile = polaimage
#mode = 'pola'
#immath()

#Call as function instead:
immath(imagenname=[qpolimage,upolimage],mode='poli',outfile=poliimage,sigma='0.0Jy/beam')
print "Created POLI image "+poliimage
immath(imagenname=[qpolimage,upolimage],mode='pola',outfile=polaimage,sigma='0.0Jy/beam')
print "Created POLA image "+polaimage

#
#-----
# Save statistics of these images

```

```

default('imstat')

imagename = poliimage
stokes = ''
box = onbox
on_statistics['POLI'] = imstat()
box = offbox
off_statistics['POLI'] = imstat()

#
#
#-----
# Display clean I image in viewer but with polarization vectors
#
# If you did not do interactive clean, bring up viewer manually
viewer(poliimage)

print "Displaying pol I now. You should overlay pola vectors"
print "Bring up the Load Data panel:"
print ""
print "Use LEL for POLA VECTOR with cut above 6*mysigma in POLI = "+str(6*mysigma)
print "For example:"
print "\'" + polaimage + "\' [\'" + poliimage + "\' > 0.0048]"
print ""
print "In the Data Display Options for the vector plot:"
print "  Set the x,y increments to 2 (default is 3)"
print "  Use an extra rotation this 90deg to get B field"
print "Note the lengths are all equal. You can fiddle these."
print ""
print "You can also load the poli image as contours"

# Pause script if you are running in scriptmode
if scriptmode:
    user_check = raw_input('Return to continue script\n')

# NOTE: the LEL will be something like
# 'jupiter6cm.demo.polimg.clean.image.pola' ['jupiter6cm.demo.polimg.clean.image.poli' > 0.005]

#
# NOTE: The viewer can take complex images to make Vector plots, although
# the image analysis tasks (and ia tool) cannot yet handle these. But we
# can use the imagepol tool (which is not imported by default) to make
# a complex image of the linear polarized intensity for display.
# See CASA User Reference Manual:
# http://casa.nrao.edu/docs/casaref/imagepol-Tool.html
#
# Make an imagepol tool and open the clean image
potool = casac.homefinder.find_home_by_name('imagepolHome')
po = potool.create()
po.open(poliimage)
# Use complexlinpol to make a Q+iU image

```

```

complexlinpolimage = polimname + '.cmlxlinpol'
po.complexlinpol(complexlinpolimage)
po.close()

# You can now display this in the viewer, in particular overlay this
# over the intensity raster with the poli contours. The vector lengths
# will be proportional to the polarized intensity. You can play with
# the Data Display Options panel for vector spacing and length.
# You will want to have this masked, like the pola image above, on
# the polarized intensity. When you load the image, use the LEL:
# 'jupiter6cm.demo.polimg.clean.cmlxlinpol'['jupiter6cm.demo.polimg.clean.image.poli'>0.005]

#=====
#
# Print results
#
print ""
print ' Jupiter polarization results '
print ' ===== '
print ''
for stokes in ['I','Q','U','V','POLI']:
    print ''
    print ' ===== '
    print ''
    print ' Polarization (Stokes '+stokes+'):'
    mymax = on_statistics[stokes]['max'][0]
    mymin = on_statistics[stokes]['min'][0]
    myrms = off_statistics[stokes]['rms'][0]
    absmax = max(mymax,mymin)
    mydra = absmax/myrms
    print ' Clean image ON-SRC max = ',mymax
    print ' Clean image ON-SRC min = ',mymin
    print ' Clean image OFF-SRC rms = ',myrms
    print ' Clean image dynamic rng = ',mydra

print '--- Done ---'

#
#=====

```

F.3 BIMA Mosaic Spectral Imaging

This script analyzes a BIMA SONG mosaic of the galaxy NGC 4826 at 3mm.

The latest version of this script can be found at:

http://casa.nrao.edu/Doc/Scripts/ngc4826_tutorial.py

```
#####
#
# Demo Script for NGC 4826 (BIMA line data)
#
# Converted by STM 2008-05-27 (Beta Patch 2.0) new tasking/clean/cal
# Updated by CB 2008-05-30 start from raw data
# Updated by STM 2008-06-01 scriptmode, plotting
# Updated by CB,STM 2008-06-02 improvements
# Updated by CB,STM 2008-06-03 bigger cube, pbcor
# Updated by CB,STM 2008-06-04 pbcor stuff
# Updated by CB,STM 2008-06-04 tutorial script
# Updated by CB 2008-06-05 small tweaks
# Updated by CB,STM 2008-06-12 final revisions (DS)
# Updated by STM 2008-06-14 post-school fix
# Updated by DP 2009-05-05 pre-Garching immoments chans
# Updated by DP 2009-05-05 messages for flagging
# Updated by CB 2009-05-18 pre-Canada spw fix not needed now
# Updated by CB 2009-05-18 immoments axis=spectral
# Updated by CB 2009-05-19 added restfreq to clean
# Updated by CB 2009-05-19 removed clearcals
# Updated by STM 2009-06-26 Patch 4, web download
# Updated by STM 2009-12-20 Release 3.0 final
#
# N4826 - BIMA SONG Data
#
# This data is from the BIMA Survey of Nearby Galaxies (BIMA SONG)
# Helfer, Thornley, Regan, et al., 2003, ApJS, 145, 259
# Many thanks to Michele Thornley for providing the data and description
#
# First day of observations only
#
# Script Notes:
#   o This script will automatically download the data from the web
#     if it cannot find the data on disk.
#   o The "default" commands are not necessary, but are included
#     in case we want to change from function calls to globals
#   o This script has some interactive commands, such as with plotxy
#     and the viewer. This script will stop and require a
#     carriage-return to continue at these points.
#   o Sometimes cut-and-paste of a series of lines from this script
#     into the casapy terminal will get garbled (usually a single
#     dropped character). In this case, try fewer lines, like groups
#     of 4-6.
#   o If you want to re-run the script, quit casapy and start again
#
#####
import os

# needed for existence check
from os import F_OK
```

```

# Might need these if having problems with plotxy windows
#pl.ion()
#pl.clf()
#
#####
#
# Clear out previous run results
#rmtables('ngc4826.tutorial.*')
os.system('rm -rf ngc4826.tutorial.*')

# Sets a shorthand for the ms, not necessary
prefix='ngc4826.tutorial'
msfile = prefix + '.16apr98.ms'

print 'Tutorial Script for BIMASONG NGC4826 Mosaic'
print 'Version for Release 0 (3.0.0) 20-Dec-2009'
print 'Will do: import, flagging, calibration, imaging'
print ''
#
#####
#
#
#####
#
# N4826 - BIMA SONG Data CO(1-0) 115.2712 GHz
# 16apr98
# source=ngc4826
# phasecal=1310+323
# fluxcal=3c273, Flux = 23 Jy on 16apr98
# passcal= none - data were observed with online bandpass correction.
#
# NOTE: This data has been filled into MIRIAD, line-length correction
# done, and then exported as separate files for each source.
# 3c273 was not line length corrected since it was observed
# for such a short amount of time that it did not need it.
#
# From miriad: source Vlsr = 408; delta V is 20 km/s
#
#####
# Locate the data
#####
#
fitsdir = 'fitsfiles'
# See if this sub-directory exists
if os.access(fitsdir,F_OK):
    # already in current directory
    print " Found "+fitsdir+" for UVFITS files"
else:
    pathname=os.environ.get('CASAPATH').split()[0]
    datapath=pathname+'/data/demo/tutorial/BIMA/NGC4826/'
    dataname='ngc4826.fitsfiles.tgz'

```

```

datafiles = datapath+dataname
# Path to web archive
webfiles = 'http://casa.nrao.edu/Data/BIMA/NGC4826/'+dataname
if os.access(dataname,F_OK):
    print '--Found data tarball in local directory--'
    print " Using "+dataname
elif os.access(datafiles,F_OK):
    print '--Copy data tarball to local directory--'
    print " Using "+datafiles
    os.system("cp -r "+datafiles+" .")
    os.system('chmod -R a+wx '+datafiles)
else:
# try web retrieval
    print '--Trying to retrieve data from '+webfiles
    # Use curl (for Mac compatibility)
    os.system('curl '+webfiles+' -f -o '+dataname)
    # NOTE: could also use wget
    #os.system('wget '+webfiles)

    print '--Unpacking tarball '
    os.system('tar xzf '+dataname)
    if os.access(fitsdir,F_OK):
        # should now be in current directory
    print " Will use "+fitsdir+" in current directory"
    else:
        raise IOError," ERROR: "+fitsdir+" not found"

#####
# Import and concatenate sources
#####
#
# USB spectral windows written separately by miriad for 16apr98
# Assumes these are in sub-directory called "fitsfiles" of working directory
print '--Importuvfits (16apr98)--'
default('importuvfits')

print "Starting from the uvfits files exported by miriad"
print "The USB spectral windows were written separately by miriad for 16apr98"

#### We could read in each of the individual fits files as example
#### below -- this works well if you only have one or two files to
#### read, but here we have many, so instead we use some useful
#### pythonease to simplify the commands.

#importuvfits(fitsfile='fitsfiles/3c273.fits5', vis='ngc4826.tutorial.3c273.5.ms')

#### Tutorial Note: For the loop to work, the high end of range must be
#### 1+ number of actual files.

for i in range(5,9):
importuvfits(fitsfile="fitsfiles/3c273.fits"+str(i),

```

```

vis="ngc4826.tutorial.3c273."+str(i)+".ms")

for i in range(9,17):
importuvfits(fitsfile="fitsfiles/1310+323.ll.fits"+str(i),
vis="ngc4826.tutorial.1310+323.ll."+str(i)+".ms")

for i in range(5,9):
importuvfits(fitsfile="fitsfiles/ngc4826.ll.fits"+str(i),
vis="ngc4826.tutorial.ngc4826.ll."+str(i)+".ms")
#
#####
#
print '--Concat--'
default('concat')

concat(vis=['ngc4826.tutorial.3c273.5.ms',
'ngc4826.tutorial.3c273.6.ms',
'ngc4826.tutorial.3c273.7.ms',
'ngc4826.tutorial.3c273.8.ms',
'ngc4826.tutorial.1310+323.ll.9.ms',
'ngc4826.tutorial.1310+323.ll.10.ms',
'ngc4826.tutorial.1310+323.ll.11.ms',
'ngc4826.tutorial.1310+323.ll.12.ms',
'ngc4826.tutorial.1310+323.ll.13.ms',
'ngc4826.tutorial.1310+323.ll.14.ms',
'ngc4826.tutorial.1310+323.ll.15.ms',
'ngc4826.tutorial.1310+323.ll.16.ms',
'ngc4826.tutorial.ngc4826.ll.5.ms',
'ngc4826.tutorial.ngc4826.ll.6.ms',
'ngc4826.tutorial.ngc4826.ll.7.ms',
'ngc4826.tutorial.ngc4826.ll.8.ms'],
concatvis='ngc4826.tutorial.ms',
freqtol="",dirtol="1arcsec",async=False)

#
#####
#
# TUTORIAL NOTES:
#
# You can invoke tasks in two ways:
#
# (1) As function calls with arguments as shown above for concat and used
# extensively in this script, e.g.
#
#     task( par1=val1, par2=val2, ... )
#
# with parameters set as arguments in the call. Note that in this
# case, the global parameter values are NOT used or changed, and any
# task parameters that are not specified as arguments to the call
# will be defaulted to the task-specific default values (see the
# "help task" description).

```

```

#
# (2) By setting the values of the global parameters and then using the
# "go" command (if taskname is set) or calling the task with no
# arguments. For example:
#
#     default task
#     par1 = val1
#     par2 = val2
#     ...
#     inp
#     task()
#
# In this case, the "default" command sets the parameters to their
# task defaults, and sets the "taskname" parameter to the task to be
# run. The "inp" command displays the current values for the task
# parameters. Then the call with no arguments runs with the globals.
#
# Warning: "go" does not work inside scripts. See Cookbook.
#
# Using the concat call above as an example, we would do:
#
#default('concat')
#
#vis = ['ngc4826.tutorial.3c273.5.ms',
#       'ngc4826.tutorial.3c273.6.ms',
#       'ngc4826.tutorial.3c273.7.ms',
#       'ngc4826.tutorial.3c273.8.ms',
#       'ngc4826.tutorial.1310+323.11.9.ms',
#       'ngc4826.tutorial.1310+323.11.10.ms',
#       'ngc4826.tutorial.1310+323.11.11.ms',
#       'ngc4826.tutorial.1310+323.11.12.ms',
#       'ngc4826.tutorial.1310+323.11.13.ms',
#       'ngc4826.tutorial.1310+323.11.14.ms',
#       'ngc4826.tutorial.1310+323.11.15.ms',
#       'ngc4826.tutorial.1310+323.11.16.ms',
#       'ngc4826.tutorial.ngc4826.11.5.ms',
#       'ngc4826.tutorial.ngc4826.11.6.ms',
#       'ngc4826.tutorial.ngc4826.11.7.ms',
#       'ngc4826.tutorial.ngc4826.11.8.ms']
#
#concatvis='ngc4826.tutorial.ms'
#freqtol = ""
#dirtol = "1arcsec"
#async=False
#
#concat()
#
#####
#
# Fix up the MS

```



```

# This ensures that the rest freq will be found for all spws.
# NOTE: STILL NECESSARY
#
# print '--Fixing up spw rest frequencies in MS--'
vis='ngc4826.tutorial.ms'
tb.open(vis+'/SOURCE',nomodify=false)
spwid=tb.getcol('SPECTRAL_WINDOW_ID')
#spwid.setfield(-1,int)
# Had to do this for 64bit systems 08-Jul-2008
spwid.setfield(-1,'int32')
tb.putcol('SPECTRAL_WINDOW_ID',spwid)
tb.close()

#####
# 16 APR Calibration
#####
#
# List contents of MS
#
print '--Listobs--'
listobs(vis='ngc4826.tutorial.ms')

# Should see the listing included at the end of this script
#

print "There are 3 fields observed in a total of 16 spectral windows"
print "  field=0    3c273    spwids 0,1,2,3           64 chans "
print "  field=1    1310+323 spwids 4,5,6,7,8,9,10,11   32 chans "
print "  field=2~8   NGC4826  spwids 12,13,14,15         64 chans "
print ""
print "See listobs summary in logger"
#
#####
# Plotting and Flagging
#####
#
# The plotxy task is the interactive x-y display and flagging GUI
#
print '--Plotxy--'
default(plotxy)

# Here we will suggest things to plot, and actually only do a few
# (near the end of this task block).  If you like you can
# uncomment these when you run this script
#
# First look at amplitude as a function of uv-distance using an
# average over all channels for each source.
#
# Interactive plotxy
#plotxy(vis='ngc4826.tutorial.ms',xaxis='uvdist',yaxis='amp',field='0',spw='0~3',
#       averagemode='vector',width='1000',selectplot=True,title='Field 0 SPW 0~3')

```

```

#
# Pause script if you are running in scriptmode
#user_check=raw_input('Return to continue script\n')

# NOTE: width here needs to be larger than combination of all channels
# selected with spw and/or field. Since field and spw are unique in this
# case, both don't need to be specified, however plotting is much faster
# if you "help it" by selecting both.
#
# Now average over all times across scans but not over channel and
# plot versus channel. There are four 64-channel spws set end-to-end
# by plotxy You should see bad edge channels in each spw segment We
# will flag these (non-interactively) later
#
# Interactive plotxy
#plotxy(vis='ngc4826.tutorial.ms',xaxis='channel',yaxis='amp',field='0',spw='0~3',
#       averagemode='vector',timebin='1e7',crossscans=True,selectplot=True,
#       newplot=False,title='Field 0 SPW 0~3')
#
# Pause script if you are running in scriptmode
#user_check=raw_input('Return to continue script\n')

# You can also plot versus velocity by setting xaxis='velocity'
#
# You might do this for all the other spw/field combos
#

# You can also non-interactively plotxy to a file
# (note this works only if iteration is not set)
# Also, see how we use Python variables to make this easier
#
#field = '0'
#spw = '0~3'
#figfile = 'ngc4826.tutorial.ms' + '.plotxy.'+field+'.spectrum.raw.png'
#plotxy(vis='ngc4826.tutorial.ms',xaxis='velocity',yaxis='amp',field=field,spw=spw,
#       averagemode='vector',timebin='1e7',crossscans=True,selectplot=True,
#       newplot=False,title='Field '+field+' SPW '+spw,interactive=False,figfile=figfile)

# Now lets look at the target source, the first of the NGC4826 mosaic fields
# which are 2~8 in this MS.
#
# Since we are plotting versus velocity we can clearly see the bad edge
# channels and the overlap between spw
#
# There is nothing terribly wrong with this data and again we will flag the
# edge channels non-interactively later for consistency.
#
# Normally, if there were obviously bad data, you would flag it here
# before calibration. To do this, hit the Mark Region button, then draw a box around
# some of the moderately high outliers, and then Flag.
#

```

```

# But this data is relatively clean, and flagging will not improve results.
#
# Interactive plotxy
plotxy(vis='ngc4826.tutorial.ms',xaxis='velocity',yaxis='amp',field='2',spw='12~15',
       averagemode='vector',timebin='1e7',crossscans=True,
       selectplot=True,newplot=False,title='Field 2 SPW 12~15')

print "You could Mark Region around outliers and Flag"
# Pause script if you are running in scriptmode
user_check=raw_input('Return to continue script\n')

#
# You could set up a Python loop to do all the N4826 fields, like this:
#
#for fld in range(2,9):
# field = str(fld)
# plotxy(vis='ngc4826.tutorial.ms',xaxis='velocity',yaxis='amp',field=field,spw=spw,
#       averagemode='vector',timebin='1e7',crossscans=True,selectplot=True,
#       newplot=False,title='Field '+field+' SPW '+spw)
#
# print "Nominally, Mark Region around outliers and Flag"
# # Pause script if you are running in scriptmode
# user_check=raw_input('Return to continue script\n')

# Back to first field.
# You can also have it iterate over baselines, using Next to advance
# Ignore baselines 1:1, 2:2 etc. as they would correspond to autocorrelations
# if they were present (they are not in this dataset)
#
# Interactive plotxy
#plotxy(vis='ngc4826.tutorial.ms',xaxis='channel',yaxis='amp',field='0',spw='0~3',
#       averagemode='vector',timebin='1e7',crossscans=True,iteration='baseline',
#       selectplot=True,newplot=False,title='Field 0 SPW 0~3')
#
# Pause script if you are running in scriptmode
#user_check=raw_input('Return to continue script\n')

#
# Finally, look for bad data. Here we look at field 8 w/o averaging
plotxy(vis='ngc4826.tutorial.ms',xaxis='time',yaxis='amp',field='8',spw='12~15',
       selectplot=True,newplot=False,title='Field 8 SPW 12~15')

print "You can see some bad data here"
print "Mark Region and Locate, look in logger"
print "This is a correlator glitch in baseline 3-9 at 06:19:30"
print "PLEASE DON'T FLAG ANYTHING HERE. THE SCRIPT WILL DO IT!"
print "In a normal session you could Mark Region and Flag."
print "Here we will use flagdata instead."
# Pause script if you are running in scriptmode
user_check=raw_input('Return to continue script\n')

```

```

# If you change xaxis='channel' you see its all channels
#
#####
#
# Flag end channels
#
print '--Flagdata--'
default('flagdata')

print ""
print "Flagging edge channels in all spw"
print " 0~3:0~1;62~63 , 4~11:0~1;30~31, 12~15:0~1;62~63 "
print ""

flagdata(vis='ngc4826.tutorial.ms', mode='manualflag',
         spw='0~3:0;1;62;63,4~11:0;1;30;31,12~15:0;1;62;63')

#
# Flag correlator glitch
#
print ""
print "Flagging bad correlator field 8 antenna 3&9 spw 15 all channels"
print " timerange 1998/04/16/06:19:00.0~1998/04/16/06:20:00.0"
print ""

flagdata(vis='ngc4826.tutorial.ms', mode='manualflag', field='8', spw='15', antenna='3&9',
         timerange='1998/04/16/06:19:00.0~1998/04/16/06:20:00.0')

#
#####
#
# Some example clean-up editing
# Slightly high almost-edge channel in field='1', spw='4' (channel 2)
# can be flagged interactively with plotxy.
#
#plotxy(vis='ngc4826.tutorial.ms',
#       xaxis='channel',yaxis='amp',field='1',spw='4',
#       averagemode='vector',timebin='1e7',crossscans=True,
#       selectplot=True,newplot=False,title='Field 1 SPW 4')

print "Completed pre-calibration flagging"

#
#####
#
# Use Flagmanager to save a copy of the flags so far
#
print '--Flagmanager--'
default('flagmanager')

```

```

print "Now will use flagmanager to save a copy of the flags we just made"
print "These are named myflags"

flagmanager(vis='ngc4826.tutorial.ms',mode='save',versionname='myflags',
            comment='My flags',merge='replace')

# Can also use Flagmanager to list all saved versions
#
#flagmanager(vis='ngc4826.tutorial.ms',mode='list')

#
#####
#
# CALIBRATION
#
#####
#
# Bandpasses are very flat because of observing mode used (online bandpass
# correction) so bandpass calibration is unnecessary for these data.
#
#####
#
# Derive gain calibration solutions.
# We will use VLA-like G (per-scan) calibration:
#
#####
#
# Set the flux density of 3C273 to 23 Jy
#
print '--Setjy (3C273)--'
default('setjy')

setjy(vis='ngc4826.tutorial.ms',field='0',fluxdensity=[23.0,0.,0.,0.],spw='0~3')
#
# Not really necessary to set spw but you get lots of warning messages if
# you don't
#
#####
#
# Gain calibration
#
print '--Gaincal--'
default('gaincal')

# This should be combining all spw for the two calibrators for single
# scan-based solutions

print 'Gain calibration for fields 0,1 and spw 0~11'
print 'Using solint=inf combining over spw'
print 'Output table ngc4826.tutorial.16apr98.gcal'

```

```

gaincal(vis='ngc4826.tutorial.ms', caltable='ngc4826.tutorial.16apr98.gcal',
field='0,1', spw='0~11', gaintype='G', minsnr=2.0,
refant='ANT5', gaincurve=False, opacity=0.0,
solint='inf', combine='spw')

#
#####
#
# Transfer the flux density scale:
#
print '--Fluxscale--'
default('fluxscale')

print ''
print 'Transferring flux of 3C273 to sources: 1310+323'
print 'Output table ngc4826.tutorial.16apr98.fcal'

fluxscale(vis='ngc4826.tutorial.ms', caltable='ngc4826.tutorial.16apr98.gcal',
fluxtable='ngc4826.tutorial.16apr98.fcal',
reference='3C273', transfer=['1310+323'])

# Flux density for 1310+323 is: 1.48 +/- 0.016 (SNR = 90.6, nAnt= 8)
#
#####
#
# Plot calibration
print '--Plotcal (fluxscale)--'
default(plotcal)

# Interactive plotcal
plotcal(caltable='ngc4826.tutorial.16apr98.fcal', yaxis='amp', field='')
print ''
print 'Plotting final scaled gain calibration table'
print 'First amp vs. time for all fields '

# Pause script if you are running in scriptmode
user_check=raw_input('Return to continue script\n')

plotcal(caltable='ngc4826.tutorial.16apr98.fcal', yaxis='phase', field='')
print ''
print 'and phase vs. time '

# Pause script if you are running in scriptmode
user_check=raw_input('Return to continue script\n')

# And you can plot the SNR of the solution
#plotcal(caltable='ngc4826.tutorial.16apr98.fcal', yaxis='snr', field='')

# You can also plotcal to file
#figfile = 'ngc4826.tutorial.16apr98.fcal.plotcal.amp.png'
#plotcal(caltable='ngc4826.tutorial.16apr98.fcal', yaxis='amp', field='', showgui=False, figfile=figfile)

```

```

#figfile = 'ngc4826.tutorial.16apr98.fcal.plotcal.phase.png'
#plotcal(caltable='ngc4826.tutorial.16apr98.fcal',yaxis='phase',field='',showgui=False,figfile=figfile)

#
#####
#
# Correct the calibrator/target source data:
# Use new parm spwmap to apply gain solutions derived from spwid1
# to all other spwids...
print '--Applycal--'
default('applycal')

print 'Applying calibration table ngc4826.tutorial.16apr98.fcal to data'

applycal(vis='ngc4826.tutorial.ms',
         field='', spw='',
         gaincurve=False, opacity=0.0,
         gaintable='ngc4826.tutorial.16apr98.fcal',
         spwmap=[0,0,0,0,0,0,0,0,0,0,0,0,0,0,0,0])

#
#####
#
# Check calibrated data
print '--Plotxy--'
default(plotxy)

#
# Here we plot the first of the NGC4826 fields unaveraged versus velocity
# Notice how the spw fit together

# Interactive plotxy
#print "Will plot all the NGC4826 calibrated data unaveraged - this will take a while"
#plotxy(vis='ngc4826.tutorial.ms',xaxis='velocity',yaxis='amp',field='2~8',spw='12~15',
#       averagemode='vector',datacolumn='corrected',selectplot=True,
#       newplot=False,title='Field 2~8 SPW 12~15')

#print ""
#print "Look for outliers, flag them if there are any bad ones"
#print ""

# Pause script if you are running in scriptmode
#user_check=raw_input('Return to continue script\n')

# You can also plot all the N4826 fields 2 through 8, for example using a loop:

#for fld in range(2,9):
# field = str(fld)
# plotxy(vis='ngc4826.tutorial.ms',xaxis='velocity',yaxis='amp',field=field,spw='11~15',
#       averagemode='vector',datacolumn='corrected',selectplot=True,
#       newplot=False,title='Field '+field+' SPW 11~15')

```

```

#
# user_check=raw_input('Return to continue script\n')

# Now here we time-average the data, plotting versus velocity

#plotxy(vis='ngc4826.tutorial.ms',xaxis='velocity',yaxis='amp',field=field,spw=spw,
#        averagemode='vector',datacolumn='corrected',timebin='1e7',crossscans=True,
#        plotcolor='blue',selectplot=True,newplot=False,title='Field '+field+' SPW '+spw)
#print ""
#print 'Final Spectrum field '+field+' spw '+spw+' TimeAverage Corrected Data'

# Pause script if you are running in scriptmode
#user_check=raw_input('Return to continue script\n')

# Here we overplot 3C273 the Time+Chan averaged calibrated and uncalibrated data

#
# First the corrected column in blue
#field = '0'
#spw = '0~3'
#plotxy(vis='ngc4826.tutorial.ms',xaxis='uvdist',yaxis='amp',field=field,spw=spw,
#        averagemode='vector',width='1000',datacolumn='corrected',timebin='1e7',
#        crossscans=True,plotcolor='blue',selectplot=True,newplot=False,
#        title='Field '+field+' SPW '+spw)
#print ""
#print 'Plotting field '+field+' spw '+spw+' TimeChanAverage Corrected Data in blue'
#
# Now the original data column in red
#plotxy(vis='ngc4826.tutorial.ms',xaxis='uvdist',yaxis='amp',field=field,spw=spw,
#        averagemode='vector',width='1000',datacolumn='data',timebin='1e7',
#        crossscans=True,plotcolor='red',overplot=True,selectplot=True,
#        newplot=False,title='Field '+field+' SPW '+spw)
#
#print 'OverPlotting field '+field+' spw '+spw+' TimeChanAverage Original Data in red'

## Pause script if you are running in scriptmode
#user_check=raw_input('Return to continue script\n')

# Can repeat for
#field = '1'
#spw = '4~11'

print "Done calibration and plotting"
#
#####
#
# SPLIT THE DATA INTO SINGLE-SOURCE MS
# AND THEN IMAGE THE CALIBRATOR
#
#####
#

```



```

#
# Split out calibrated target source and calibrator data:
#
print '--Split--'
default('split')

print 'Splitting 3C273 data to ngc4826.tutorial.16apr98.3C273.split.ms'

split(vis='ngc4826.tutorial.ms',
      outputvis='ngc4826.tutorial.16apr98.3C273.split.ms',
      field='0', spw='0~3:0~63', datacolumn='corrected')

print 'Splitting 1310+323 data to ngc4826.tutorial.16apr98.1310+323.split.ms'

split(vis='ngc4826.tutorial.ms',
      outputvis='ngc4826.tutorial.16apr98.1310+323.split.ms',
      field='1', spw='4~11:0~31', datacolumn='corrected')

print 'Splitting NGC4826 data to ngc4826.tutorial.16apr98.src.split.ms'

split(vis='ngc4826.tutorial.ms',
      outputvis='ngc4826.tutorial.16apr98.src.split.ms',
      field='2~8', spw='12~15:0~63',
      datacolumn='corrected')

#
#####
#
# If you want to use plotxy before cleaning to look at the split ms
#plotxy(vis='ngc4826.tutorial.16apr98.src.split.ms',xaxis='time',yaxis='amp')
#
#####
#
# You might image the calibrator data:
#
#print '--Clean (1310+323)--'
#default('clean')
#
#
#clean(vis='ngc4826.tutorial.16apr98.1310+323.split.ms',
#      imagename='ngc4826.tutorial.16apr98.cal.clean',
#      cell=[1.,1.], imsize=[256,256], field='0', spw='0~7',
#      threshold=10., mode='mfs', psfmode='clark', niter=100, stokes='I')

# You can look at this in the viewer
#viewer('ngc4826.tutorial.16apr98.cal.clean.image')

#
#
#####
#

```

```

# IMAGING OF NGC4826 MOSAIC
#
#####
#
#       Mosaic field spacing looks like:
#
#       F3 (field 3)           F2 (field 2)
#
#   F4 (field 4)           F0 (field 0)           F1 (field 1)
#
#       F5 (field 5)           F6 (field 6)
#
# 4x64 channels = 256 channels
#
# Primary Beam should be about 1.6' FWHM (7m dishes, 2.7mm wavelength)
# Resolution should be about 5-8"
#####
#
# Image the target source mosaic:
#
print '--Clean (NGC4826)--'
default('clean')

clean(vis='ngc4826.tutorial.16apr98.src.split.ms',
      imagename='ngc4826.tutorial.16apr98.src.clean',
      field='0~6',spw='0~3',
      cell=[1.,1.],imsize=[256,256],stokes='I',
      mode='channel',nchan=36,start=35,width=4,
      interpolation='nearest',
      psfmode='clark',imagermode='mosaic',scaletype='SAULT',
      cyclefactor=4,ftmachine='mosaic',
      niter=10000,threshold='45mJy',
      restfreq='115.2712GHz',interactive=F,
      minpb=0.3,pbcor=False)

### NOTE: Interpolation = 'nearest', can try 'linear' also

### NOTE: Explicitly set restfreq to CO 1-0

### NOTE: Sault weighting implies a uniform noise mosaic

### NOTE: that niter is set to large number so that stopping point is
### controlled by threshold.

### NOTE: with pbcor=False, the final image is not "flux correct",
### instead the image has constant noise despite roll off in power as
### you move out from the phase center(s). Though this format makes it
### "look nicest", for all flux density measurements, and to get an
### accurate integrated intensity image, one needs to divide the
### srcimage.image/srcimage.flux in order to correct for the mosaic
### response pattern. One could also achieve this by setting pbcor=True

```

```

### in clean.

# Try running clean adding the parameter interactive=True.
# This parameter will periodically bring up the viewer to allow
# interactive clean boxing. For poor uv-coverage, deep negative bowls
# from missing short spacings, this can be very important to get correct
# integrated flux densities.

#
#####
#
# Do interactive viewing of clean image
print '--Viewer--'
viewer('ngc4826.tutorial.16apr98.src.clean.image')

print ""
print "This is the non-pbcorrected cube of NGC4826"
print "Use tape deck to move through channels"
print "Close the viewer when done"
print ""
#
# Pause script if you are running in scriptmode
user_check=raw_input('Return to continue script\n')

#
#####
#
# Statistics on clean image cube
#
print '--ImStat (Clean cube)--'

srcstat = imstat('ngc4826.tutorial.16apr98.src.clean.image')

print "Found image max = "+str(srcstat['max'])[0]

# offbox = '106,161,153,200'

offstat = imstat('ngc4826.tutorial.16apr98.src.clean.image',
                 box='106,161,153,200')

print "Found off-source image rms = "+str(offstat['sigma'])[0]

# cenbox = '108,108,148,148'
# offlinechan = '0,1,2,3,4,5,30,31,32,33,34,35'

offlinestat = imstat('ngc4826.tutorial.16apr98.src.clean.image',
                     box='108,108,148,148',
                     chans='0,1,2,3,4,5,30,31,32,33,34,35')

print "Found off-line image rms = "+str(offlinestat['sigma'])[0]

```

```

#
#####
#
# Manually correct for mosaic response pattern using .image/.flux images
#
print '--ImMath (PBcor)--'

immath(outfile='ngc4826.tutorial.16apr98.src.clean.pbcor',
        imagename=['ngc4826.tutorial.16apr98.src.clean.image', 'ngc4826.tutorial.16apr98.src.clean.flux'],
        mode='evalexpr', expr='IM0/IM1')

#
#####
#
# Statistics on PBcor image cube
#
print '--ImStat (PBcor cube)--'

pbcorstat = imstat('ngc4826.tutorial.16apr98.src.clean.pbcor')

print "Found image max = "+str(pbcorstat['max'][0])

pbcoroffstat = imstat('ngc4826.tutorial.16apr98.src.clean.pbcor',
                      box='106,161,153,200')

print "Found off-source image rms = "+str(pbcoroffstat['sigma'][0])

pbcorofflinestat = imstat('ngc4826.tutorial.16apr98.src.clean.pbcor',
                           box='108,108,148,148',
                           chans='0,1,2,3,4,5,30,31,32,33,34,35')

print "Found off-line image rms = "+str(pbcorofflinestat['sigma'][0])

#
#####
#
# Do zeroth and first moments
#
# NGC4826 LSR velocity is 408 km/s; delta is 20 km/s
#
print '--ImMoments--'
default('immoments')

momfile = 'ngc4826.tutorial.16apr98.moments'
momzeroimage = 'ngc4826.tutorial.16apr98.moments.integrated'
momoneimage = 'ngc4826.tutorial.16apr98.moments.mom1'

print "Calculating Moments 0,1 for PBcor image"

immoments(imagename='ngc4826.tutorial.16apr98.src.clean.pbcor',
           moments=0,axis='spectral',

```

```

chans='7~28',
      outfile='ngc4826.tutorial.16apr98.moments.integrated')

# TUTORIAL NOTES: For moment 0 we use the image corrected for the
# mosaic response to get correct integrated flux densities. However,
# in *real signal* regions, the value of moment 1 does not depend on
# the flux being correct so the non-pb corrected SAULT image can be
# used, this avoids having lots of junk show up at the edges of your
# moment 1 image due to the primary beam correction. Try it both ways
# and see for yourself.

# TUTORIAL NOTES:
#
# Moments greater than zero need to have a conservative lower
# flux cutoff to produce sensible results.

immoments(imagename='ngc4826.tutorial.16apr98.src.clean.image',
          moments=1,axis='spectral',includepix=[0.2,1000.0],
          chans='7~28',
          outfile='ngc4826.tutorial.16apr98.moments.mom1')

# Now view the resulting images
viewer('ngc4826.tutorial.16apr98.moments.integrated')
#
print "Now viewing Moment-0 ngc4826.tutorial.16apr98.moments.integrated"
print "Note PBCOR effects at field edge"
print "Change the colorscale to get better image"
print "You can also Open and overlay Contours of Moment-1 ngc4826.tutorial.16apr98.moments.mom1"
print "Close the viewer when done"
#
# Pause script if you are running in scriptmode
user_check=raw_input('Return to continue script\n')

#
#####
#
# Statistics on moment images
#
print '--ImStat (Moment images)--'

momzerostat=imstat('ngc4826.tutorial.16apr98.moments.integrated')

print "Found moment 0 max = "+str(momzerostat['max'])[0])

print "Found moment 0 rms = "+str(momzerostat['rms'])[0])

momonestat=imstat('ngc4826.tutorial.16apr98.moments.mom1')

print "Found moment 1 median = "+str(momonestat['median'])[0])

#

```

```
#####
#
# An alternative is to mask the pbcor image before calculating
# moments. The following block shows how to do this.

#print '--Viewer--'
#viewer(ngc4826.tutorial.16apr98.moments.integrated)

# TUTORIAL NOTES: After loading change "unit contour level" in "Data
# Display Options" gui to something like 70. select "region manager
# tool" from "tool" drop down menu Then assign the sqiggly Polygon
# button to a mouse button by clicking on it with a mouse button. Then
# draw a polygon region around galaxy emission, avoiding edge regions
# and double click in the region you created. Then in "region manager
# tool" select "save to file" and give file name mom0mask.rgn.

#print '--ImMoments (masked)--'
#print 'Creating masked moment 0 image ngc4826.tutorial.16apr98.moments.integratedmasked'
#
#immoments(imagename='ngc4826.tutorial.16apr98.src.clean.pbcor',moments=0,
#          axis='spectral',chans='7~28',region='mom0mask.rgn',
#          outfile='ngc4826.tutorial.16apr98.moments.integratedmasked')
#
#print 'Creating masked moment 1 image ngc4826.tutorial.16apr98.moments.mom1masked'
#
#immoments(imagename='ngc4826.tutorial.16apr98.src.clean.pbcor.masked',moments=1,
#          axis='spectral',includepix=[0.2,1000.0],chans='7~28',
#          region='mom0mask.rgn',outfile='ngc4826.tutorial.16apr98.moments.mom1masked')

# Now view the resulting images
#viewer('ngc4826.tutorial.16apr98.moments.integratedmasked')
#
#print "Now viewing masked Moment-0 ngc4826.tutorial.16apr98.moments.integratedmasked"
#print "You can Open and overlay Contours of Moment-1 ngc4826.tutorial.16apr98.moments.mom1masked"
#
# Pause script if you are running in scriptmode
#user_check=raw_input('Return to continue script\n')

# Finally, can compute and print statistics
#print '--ImStat (masked moments)--'
#
#maskedmomzerostat = imstat('ngc4826.tutorial.16apr98.moments.integratedmasked')
#print "Found masked moment 0 max = "+str(maskedmomzerostat['max'][0])
#print "Found masked moment 0 rms = "+str(maskedmomzerostat['rms'][0])
#
#maskedmomonestat=imstat('ngc4826.tutorial.16apr98.moments.mom1masked')
#print "Found masked moment 1 median = "+str(maskedmomonestat['median'][0])

#
#####
#
```

```

# Now show how to print out results
#
print '--Results (16apr98)--'
print ''
#
# Currently using non-PBcor values
#
im_srcmax16 = srcstat['max'][0]
im_offrms16 = offstat['sigma'][0]
im_offlinerms16 = offlinestat['sigma'][0]
thistest_immax = momzerostat['max'][0]
thistest_imrms = momzerostat['rms'][0]

#
# Report a few key stats
#
print '  NGC4826 Image Cube Max = '+str(im_srcmax16)
print "      At ("+str(srcstat['maxpos'][0])+","+str(srcstat['maxpos'][1])+") Channel "+str(srcstat[
print '      '+srcstat['maxposf']
print ''
print '      Off-Source Rms = '+str(im_offrms16)
print '      Signal-to-Noise ratio = '+str(im_srcmax16/im_offrms16)
print ''
print '      Off-Line   Rms = '+str(im_offlinerms16)
print '      Signal-to-Noise ratio = '+str(im_srcmax16/im_offlinerms16)

print "Done with NGC4826 Tutorial"
#
#####

```

Appendix G

CASA Dictionaries

BETA ALERT: These tend to become out of date as we add new tasks or change names.

G.1 AIPS – CASA dictionary

Please see:

- <https://wikio.nrao.edu/bin/view/Software/CASA-AIPSDictionary>

BETA ALERT: This link is out-of-date and refers mostly to the Toolkit. We will update this with a task dictionary.

G.2 MIRIAD – CASA dictionary

Table G.1 provides a list of common Miriad tasks, and their equivalent CASA tool or tool function names. The two packages differ in both their architecture and calibration and imaging models, and there is often not a direct correspondence. However, this index does provide a scientific user of CASA who is familiar with MIRIAD, with a simple translation table to map their existing data reduction knowledge to the new package.

G.3 CLIC – CASA dictionary

Table G.2 provides a list of common CLIC tasks, and their equivalent CASA tool or tool function names. The two packages are very similar since the CASA software to reduce IRAM data is based on the CLIC reduction procedures.

Table G.1: MIRIAD – CASA dictionary

MIRIAD Task	Description	CASA task/tool
atlod	load ATCA data	atcafiller tool
blflag	Interactive baseline based editor/flagger	mp raster displays
cgcurs	Interactive image analysis	viewer
cgdisp	Image display, overlays	viewer
clean	Clean an image	clean
fits	FITS image filler	importfits
gpboot	Set flux density scale	fluxscale
gpcal	Polarization leakage and gain calibration	cb with 'G' and 'D'
gpcopy	copy calibration tables	<i>not needed</i>
gpplt	Plot calibration solutions	plotcal
incomb	Image combination	im tool
imfit	Image-plane component fitter	ia.imagefitter
impol	Create polarization images	ia.imagepol
imstat	Image statistics	ia.statistics
imsub	Extract sub-image	ia.subimage
invert	Synthesis imaging	invert, im tool
linmos	linear mosaic combination of images	mosaic
maths	Calculations involving images	ia.imagecalc, ia.calc
mfcad	Bandpass and gain calibration	bandpass
prthd	Print header of image or uvdata	imhead, listobs
restor	Restore a clean component model	im tool
selfcal	selfcalibration of visibility data	clean, gaincal, etc.
tvclip	automated flagging based on clip levels	flagdata
tvdisp	Load image to TV display	viewer
tvflag	Interactive TB data editing	viewer
uvaver	Average/select data, apply calibration	applycal, split
uvfit	uv-plane component fitter	uvmodelfit
uvflag	Command-based flagging	flagdata
uvgen	Simulator	sm tool
uvlist	List uv-data	listvis (TBD)
uvmodel	Source model computation	ft
uvplt	uv-data plotting	plotxy
uvsplit	split uv file in sources and spectral windows	split

Table G.2: CLIC–CASA dictionary

CLIC Function	Description	CASA task/tool
load	Load data	almatifiller tool
print	Print text summary of data	listobs
flag	Flag data	plotxy, flagdata, viewer
phcor	Atmospheric phase correction	almatifiller
rf	Radio frequency bandpass	bandpass
phase	Phase calibration	gaincal
flux	Absolute flux calibration	setjy, fluxscale
ampl	Amplitude calibration	gaincal
table	Split out calibrated data (uv table)	split

Appendix H

Writing Tasks

ALERT: This prescription for writing and incorporating tasks in CASA is for the power-user. This procedure is also likely to change in future releases.

It is possible to write your own task and have it appear in casapy. For example, if you want to create a task named “yourtask”, then must create two files, `yourtask.xml` and a `task_yourtask.py`. The `.xml` file is use to describe the interface to the task and the `task_yourtask.py` does the actual work. The argument names must be the same in both the `yourtask.xml` and `task_yourtask.py` file. The `yourtask.xml` file is used to generate all the interface files so `yourtask` will appear in the casapy system. It is easiest to start from one of the existing tasks when constructing these. You would make the name of the function in the `yourtask.py` be “yourtask” in this example.

We have provided the `buildmytasks` command (outside casa) in order to assemble your Python and XML into a loadable Python file. Thus, the steps you need to execute (again for an example task named “yourtask”):

- Create python code for task as `yourtask_task.py`
- Create xml for task as `yourtask.xml`
- Execute outside casapy: `buildmytasks yourtask`
- Execute inside casapy: `execfile 'mytasks.py'`

After this, you should see the help and inputs inside casapy, e.g. `inp yourtask` should work. Note that for the final step you invoke the file called `mytasks.py`, regardless of what you named the actual task.

Note that in order to run the `buildmytasks` command in your OS, you need the path to that executable. This is set up if you’ve sourced one of the `casainit.sh` files, otherwise you will have to find this on your system,

```
/usr/lib64/casapy/24.0.8115-001/bin/buildmytasks yourtask
```

for the 2.4.0 beta release on Linux at the AOC. Note also if the files you have provided are not in the directory you are working from in CASA, you would provide that path

```
execfile '<MYPATH>/mytasks.py'
```

where <MYPATH> is that path.

ALERT: This has changed since Beta Patch 2.4 in that before you would do `execfile 'yourtask.py'` instead of `execfile 'mytasks.py'`.

H.1 The XML file

The key to getting your task into casapy is constructing a task interface description XML file.

Some XML basics, an xml element begins with <element> and ends with </element>. If an XML element contains no other XML element you may specify it via <element/>. An XML element may have zero or more attributes which are specified by attribute="attribute value". You must put the attribute value in quotes, i.e. <element myattribute="attribute value">.

All task xml files must start with this header information.

```
<?xml version="1.0" encoding="UTF-8"?>
<?xml-stylesheet type="text/xsl" ?>
<casaxml xmlns="http://casa.nrao.edu/schema/psetTypes.html"
xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
xsi:schemaLocation="http://casa.nrao.edu/schema/casa.xsd
file:///opt/casa/code/xmlcasa/xml/casa.xsd">
```

and the file must have the end tag

```
</casaxml>
```

Inside a <task> tags you will need to specify the following elements.

<task>

Attributes

type required, allowed value is "function"

name required

Subelements

shortdescription required

description required

input optional
output optional
returns optional
constraints optional

<**shortdescription**> - required by <task>; A short one-line description describing your task

Attributes

None

Subelements

None

<**description**> - required] by <task>, Also used by <param>a; A longer description describing your task with multiple lines

Attributes

None

Subelements

None

<**input**> - optional element used by <task>; An input block specifies which parameters are used for input

Attributes

None

Subelements

<**param**> , optional

<**output**> - **optional** An output element that contains a list of parameters that are "returned" by the task.

Attributes

None

Subelements

<**param**> , optional

<**returns**> - **optional** Value returned by the task

Attributes

type optional; as specified in <param>

Subelements

<**description**> , optional

<**constraints**> - **optional** A constraints element that lets you constrain params based on the values of other params.

Attributes

None

Subelements

<when> , required.

<param> - **optional** The input and output elements consist of param elements.**Attributes****type** , required; allowed values are record, variant, string int, double, bool, intArray, doubleArray, boolArray, stringArray**name** , required;**subparam** , optional; allowed values True, False, Yes or No.**kind** , optional;**mustexist** , optional; allowed values True, False, Yes or No.

All param elements require name and type attributes.

Subelements

<description> , required;

<value> , optional;

<allowed> , optional;

<value> - **optional** Value returned by the task**Attributes****type** , required; as specified in <param> attributes.**Subelements**

<value> , optional

<allowed> - optional; Block of allowed values

Attributes**enum** , required; maybe enum or range. If specified as enum only specific values are allowed If specified as range then the value tags may have min and max attributes.**Subelements**

<value> , optional

<when> - **optional** When blocks allow value specific handling for parameters**Attributes****param** , required; Specifies special handling for a <param>**Subelements**

<equals> , optional

<notequals> , optional

<equals> - optional Reset parameters if equal to the specified value

Attributes

value , required; the value of the parameter

Subelements

<default> , required

<notequals> - optional Reset specified parameters if not equal to the specified value

Attributes

value , required; The value of the parameter

Subelements

<default> , optional

<default> - optional Resets default values for specified parameters

Attributes

param , required; Name of the <param> to be reset.

Subelements

<value> , required, the revised value of the <param>.

<example> - optional An example block, typically in python

Attributes

lang optional; specifies the language of the example, defaults to python.

Subelements

None

H.2 The task_yourtask.py file

You must write the python code that does the actual work. The task_*.py file function call sequence must be the same as specified in the XML file. We may relax the requirement that the function call sequence exactly match the sequence in the XML file in a future release.

H.3 Example: The clean task

H.3.1 File clean.xml

Clean.xml gives a fairly comprehensive example of how to construct the XML file.

```

<?xml version="1.0" encoding="UTF-8"?>
<?xml-stylesheet type="text/xsl" ?>
<casaxml xmlns="http://casa.nrao.edu/schema/psetTypes.html"
xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
xsi:schemaLocation="http://casa.nrao.edu/schema/casa.xsd
file:///opt/casa/code/xmlcasa/xml/casa.xsd">

<!-- This is the param set for clean -->
<!-- This does the equivalent of -->
<!-- imgr:=imager('anyfile.ms'); -->
<!-- imgr.setdata(mode='channel',nchan=100,start=1,step=1,fieldid=1) -->
<!-- imgr.setimage(nx=512,ny=,cellx='1arcsec',celly='1arcsec',stokes='I',-->
<!--           mode='channel',start=35,step=1,nchan=40, -->
<!--           fieldid=[1]) -->
<!-- imgr.weight('natural'); -->
<!-- imgr.clean(algorithm='csclean',niter=500,model='field1') -->

<task type="function" name="clean">

  <shortdescription>Deconvolve an image with selected algorithm</shortdescription>

  <description>
Form images from visibilities. Handles continuum and spectral line cubes.
</description>

  <input>

    <param type="string" name="vis" kind="ms" mustexist="true">
      <description>name of input visibility file</description>
      <value></value>
    </param>

    <param type="string" name="imagenam">
      <description>Pre-name of output images</description>
      <value></value>
    </param>

    <param type="string" name="field">
      <description>Field Name</description>
      <value></value>
    </param>

    <param type="any" name="spw">

```



```

<description>Spectral windows:channels: \'\ ' is all </description>
<any type="variant"/>
<value type="string"></value>
</param>
<param type="bool" name="selectdata">
<description>Other data selection parameters</description>
<value>False</value>
</param>

<param type="string" name="timerange" subparam="true">
<description>Range of time to select from data</description>
<value></value>
</param>
<param type="string" name="uvrange" subparam="true">
<description>Select data within uvrange </description>
<value></value>
</param>
<param type="string" name="antenna" subparam="true">
<description>Select data based on antenna/baseline</description>
<value></value>
</param>
<param type="string" name="scan" subparam="true">
<description>scan number range</description>
<value></value>
</param>

<param type="string" name="mode">
<description>
    Type of selection (mfs, channel, velocity, frequency)
</description>
<value>mfs</value>
<allowed kind="enum">
<value>mfs</value>
<value>channel</value>
<value>velocity</value>
<value>frequency</value>
</allowed>
</param>

<param type="int" name="niter">
<description>Maximum number of iterations</description>
<value>500</value>

```

```

</param>

<param type="double" name="gain">
<description>Loop gain for cleaning</description>
<value>0.1</value>
</param>

<param type="double" name="threshold" units="mJy">
<description>Flux level to stop cleaning. Must include units</description>
<value>0.0</value>
</param>

<!-- Getting rid of this
<param type="bool" name="csclean">
  <description>Use Cotton-Schwab style reconciliation with UV-data</description>
  <value>False</value>

</param>
-->

<param type="string" name="psfmode">
<description>method of PSF calculation to use during minor cycles</description>
<value>clark</value>
<allowed kind="enum">
  <value>clark</value>
  <value>hogbom</value>
</allowed>
</param>

<param type="string" name="imagermode">
  <description> Use csclean or mosaic. If '\', use psfmode</description>
  <value></value>
  <allowed kind="enum">
    <value></value>
    <value>csclean</value>
    <value>mosaic</value>
  </allowed>

</param>

<param type="string" name="ftmachine" subparam="true">
<description>Gridding method for the image</description>
<value>mosaic</value>
<allowed kind="enum">
  <value>mosaic</value>

```

```

    <value>ft</value>
    <value>sd</value>
    <value>both</value>
</allowed>

</param>
<param type="bool" name="mosweight" subparam="true">
  <description>Individually weight the fields of the mosaic</description>
  <value>False</value>
</param>
<param type="string" name="scaletype" subparam="true">
  <description>Controls scaling of pixels in the image plane.
    default=\`SAULT\`;
    example: scaletype=\`PBCOR\` Options: \`PBCOR\`,\`SAULT\`</description>
  <value>SAULT</value>
  <allowed kind="enum">
<value>SAULT</value>
<value>PBCOR</value>
  </allowed>
</param>

<param type="intArray" name="multiscale">
  <description>set deconvolution scales (pixels),
    default: multiscale=[] (standard CLEAN)</description>
  <value type="vector">
<value></value>
  </value>

</param>
<param type="int" name="negcomponent" subparam="true">
  <description>
    Stop cleaning if the largest scale finds this number of neg components
  </description>
  <value>0</value>
</param>

<param type="bool" name="interactive">
<description>use interactive clean (with GUI viewer)</description>
<value>False</value>
</param>

<param type="any" name="mask">
<description>cleanbox(es), mask image(s), and/or region(s) used in cleaning

```

```

        </description>
<any type="variant"/>
<value type="stringArray"></value>
</param>

<param type="int" name="nchan" subparam="true">
<description>Number of channels (planes) in output image</description>
<value>1</value>
</param>

<param type="any" name="start" subparam="true">
<description>First channel in input to use</description>
<any type="variant"/>
<value type="int">0</value>
</param>

<param type="any" name="width" subparam="true">
<description>Number of input channels to average</description>
<any type="variant"/>
<value type="int">1</value>
</param>

<param type="intArray" name="imsize">
<description>x and y image size in pixels, symmetric for single value
        </description>
<value type="vector">
<value>256</value><value>256</value>
</value>
</param>

<param type="doubleArray" name="cell" units="arcsec">
<description>x and y cell size. default unit arcsec</description>
<value type="vector"><value>1.0</value><value>1.0</value></value>
</param>

<param type="any" name="phasecenter">
<description>Image phase center: position or field index</description>
<any type="variant"/>
<value type="string"></value>
</param>

<param type="string" name="restfreq">
<description>rest frequency to assign to image (see help)</description>

```

```
<value></value>
</param>
```

```
<param type="string" name="stokes">
<description>Stokes params to image (eg I,IV, QU,IQUV)</description>
<value>I</value>
<allowed kind="enum">
<value>I</value>
<value>IV</value>
<value>QU</value>
<value>IQUV</value>
<value>RR</value>
<value>LL</value>
<value>RRL<</value>
<value>XX</value>
<value>YY</value>
<value>XXYY</value>
</allowed>
</param>
```

```
<param type="string" name="weighting">
<description>Weighting to apply to visibilities</description>
<value>natural</value>
<allowed kind="enum">
<value>natural</value>
<value>uniform</value>
<value>briggs</value>
<value>briggsabs</value>
<value>radial</value>
<value>superuniform</value>
</allowed>
</param>
```

```
<param type="double" name="robust" subparam='true'>
<description>Briggs robustness parameter</description>
<value>0.0</value>
<allowed kind="range">
<value range="min">-2.0</value>
<value range="max">2.0</value>
</allowed>
```

```

</param>

<param type="bool" name="uvtaper">
<description>Apply additional uv tapering of visibilities.</description>
<value>False</value>
</param>

<param type="stringArray" name="outertaper" subparam="true">
<description>uv-taper on outer baselines in uv-plane</description>
<value type="vector">
  <value></value>
</value>
</param>

<param type="stringArray" name="innertaper" subparam="true">
<description>uv-taper in center of uv-plane</description>
<value>1.0</value>
</param>

<param type="string" name="modelimage">
<description>Name of model image(s) to initialize cleaning</description>
<value></value>
</param>
<param type="stringArray" name="restoringbeam">
  <description>Output Gaussian restoring beam for CLEAN image</description>
  <value></value>
</param>
<param type="bool" name="pbcor">
<description>Output primary beam-corrected image</description>
<value>False</value>
</param>

<param type="double" name="minpb">
<description>Minimum PB level to use</description>
<value>0.1</value>
</param>

<param type="any" name="noise" subparam='true'>
<description>noise parameter for briggs abs mode weighting</description>
<any type="variant"/>
<value type="string">1.0Jy</value>
</param>

<param type="int" name="npixels" subparam='true'>

```

```

    <description>number of pixels for superuniform or briggs weighting
      </description>
    <value>0</value>
  </param>

  <param type="int" name="npercycle" subparam='true'>
    <description>Number of iterations before interactive prompt</description>
    <value>100</value>
  </param>
  <param type="double" name="cyclefactor" subparam='true'>
    <description>change depth in between of csclean cycle</description>
    <value>1.5</value>
  </param>
  <param type="int" name="cyclespeedup" subparam='true'>
    <description>Cycle threshold doubles in this number of iteration</description>
    <value>-1</value>
  </param>

  <constraints>
    <when param="selectdata">
      <equals type="bool" value="False"/>
      <equals type="bool" value="True">
        <default param="timerange"><value type="string"></value>
        </default>
        <default param="uvrange"><value type="string"></value>
        </default>
        <default param="antenna"><value type="string"></value>
        </default>
        <default param="scan"><value type="string"></value>
        </default>
      </equals>
    </when>
    <when param="multiscale">
      <notequals type="vector" value="[]" >
        <default param="negcomponent"><value>-1</value>
        </default>
      </notequals>
    </when>
    <when param="mode">
      <equals value="mfs"/>

```

```

<equals value="channel">
<default param="nchan"><value>1</value></default>
<default param="start"><value>0</value>
  <description>first input channel to use</description>
</default>
<default param="width"><value>1</value></default>
</equals>
  <equals value="velocity">
<default param="nchan"><value>1</value></default>
<default param="start"><value type="string">0.0km/s</value>
  <description>Velocity of first image channel: e.g \'0.0km/s\'
    </description>
</default>
  <default param="width"><value type="string">1km/s</value>
  <description>image channel width in velocity units:
    e.g \'-1.0km/s\'</description>
</default>
  </equals>
  <equals value="frequency">
<default param="nchan"><value>1</value></default>
<default param="start"><value type="string">1.4GHz</value>
  <description>Frequency of first image channel: e.q. \'1.4GHz\'
    </description>
</default>
  <default param="width"><value type="string">10kHz</value>
  <description>Image channel width in frequency units:
    e.g \'1.0kHz\'</description>
</default>
  </equals>
</when>

  <when param="weighting">
<equals value="natural"/>
  <equals value="uniform"/>
  <equals value="briggs">
<default param="robust"><value>0.0</value></default>
<default param="npixels"><value>0</value>
  <description>number of pixels to determine uv-cell size
    0=> field of view</description>
</default>
  </equals>
<equals value="briggsabs">
<default param="robust"><value>0.0</value></default>
<default param="noise"><value type="string">1.0Jy</value></default>
<default param="npixels"><value>0</value>

```



```

    <description>number of pixels to determine uv-cell size
                                0=&gt; field of view</description>
  </default>
    </equals>
    <equals value="superuniform">
<default param="npixels"><value>0</value>
  <description>number of pixels to determine uv-cell size
                                0=&gt; +/-3pixels</description>
  </default>
    </equals>
    </when>
    <when param="uvtaper">
<equals type="bool" value="False"/>
<equals type="bool" value="True">
<default param="outertaper"><value type="vector"></value></default>
<default param="innertaper"><value type="vector"></value></default>
    </equals>
    </when>
    <when param="interactive">
<equals type="bool" value="False"/>
<equals type="bool" value="True">
    <default param="npercycle"><value>100</value></default>
    </equals>
    </when>
    <when param="imagermode">
<equals value=""/>
<equals value="csclean">
<default param="cyclefactor"><value>1.5</value></default>
<default param="cyclespeedup"><value>-1</value></default>
    </equals>
        <equals value="mosaic">
<default param="mosweight"><value>False</value></default>
<default param="ftmachine"><value type="string">mosaic</value>
        </default>
<default param="scaletype"><value type="string">SAULT</value>
        </default>
<default param="cyclefactor"><value>1.5</value></default>
<default param="cyclespeedup"><value>-1</value></default>
    </equals>
    </when>
<!--Get rid of that soon
    <when param="mosaicmode">
<equals type="bool" value="False"/>
<equals type="bool" value="True">
<default param="mosweight"><value>False</value></default>

```

```

<default param="ftmachine"><value type="string">mosaic</value></default>
<default param="scaletype"><value type="string">SAULT</value></default>
  </equals>
  </when>
-->
  </constraints>

  </input>

<returns type="void"/>

```

<example>

The main clean deconvolution task. It contains many functions

- 1) Make 'dirty' image and 'dirty' beam (psf)
- 2) Multi-frequency-continuum images or spectral channel imaging
- 3) Full Stokes imaging
- 4) Mosaicking of several pointings
- 5) Multi-scale cleaning
- 6) Interactive clean boxing
- 7) Initial starting model

```

vis -- Name of input visibility file
      default: none; example: vis='ngc5921.ms'
imasename -- Pre-name of output images:
            default: none; example: imasename='m2'
            output images are:
              m2.image; cleaned and restored image
                With or without primary beam correction
              m2.psf; point-spread function (dirty beam)
              m2.flux; relative sky sensitivity over field
              m2.model; image of clean components
              m2.residual; image of residuals
              m2.interactive.mask; image containing clean regions
field -- Select fields in mosaic. Use field id(s) or field name(s).
        ['go listobs' to obtain the list id's or names]
        default: ''=all fields
        If field string is a non-negative integer, it is assumed to
          be a field index otherwise, it is assumed to be a
field name
        field='0~2'; field ids 0,1,2
        field='0,4,5~7'; field ids 0,4,5,6,7
        field='3C286,3C295'; field named 3C286 and 3C295

```

```

field = '3,4C*'; field id 3, all names starting with 4C
spw -- Select spectral window/channels
NOTE: This selects the data passed as the INPUT to mode
default: ''=all spectral windows and channels
spw='0~2,4'; spectral windows 0,1,2,4 (all channels)
spw='0:5~61'; spw 0, channels 5 to 61
spw='<2'; spectral windows less than 2 (i.e. 0,1)
spw='0,10,3:3~45'; spw 0,10 all channels, spw 3,
channels 3 to 45.
spw='0~2:2~6'; spw 0,1,2 with channels 2 through 6 in each.
spw='0:0~10;15~60'; spectral window 0 with channels
0-10,15-60
spw='0:0~10,1:20~30,2:1;2;3'; spw 0, channels 0-10,
spw 1, channels 20-30, and spw 2, channels, 1,2 and 3
selectdata -- Other data selection parameters
default: True
>>> selectdata=True expandable parameters
See help par.selectdata for more on these
timerange -- Select data based on time range:
default = '' (all); examples,
timerange = 'YYYY/MM/DD/hh:mm:ss~YYYY/MM/DD/hh:mm:ss'
Note: if YYYY/MM/DD is missing date defaults to first
day in data set
timerange='09:14:0~09:54:0' picks 40 min on first day
timerange= '25:00:00~27:30:00' picks 1 hr to 3 hr
30min on NEXT day
timerange='09:44:00' pick data within one integration
of time
timerange='>10:24:00' data after this time
uvrange -- Select data within uvrange (default units meters)
default: '' (all); example:
uvrange='0~1000klambda'; uvrange from 0-1000 kilo-lambda
uvrange='>4klambda';uvranges greater than 4 kilo lambda
antenna -- Select data based on antenna/baseline
default: '' (all)
If antenna string is a non-negative integer, it is
assumed to be an antenna index, otherwise, it is
considered an antenna name.
antenna='5&6'; baseline between antenna index 5 and
index 6.
antenna='VA05&VA06'; baseline between VLA antenna 5
and 6.
antenna='5&6;7&8'; baselines 5-6 and 7-8
antenna='5'; all baselines with antenna index 5
antenna='05'; all baselines with antenna number 05

```

(VLA old name)

```

    antenna='5,6,9'; all baselines with antennas 5,6,9
index numbers
    scan -- Scan number range.
        default: '' (all)
        example: scan='1~5'
        Check 'go listobs' to insure the scan numbers are in
order.
mode -- Frequency Specification:
    NOTE: See examples below:
    default: 'mfs'
        mode = 'mfs' means produce one image from all
specified data.
        mode = 'channel'; Use with nchan, start, width to specify
        output image cube. See examples below
        mode = 'velocity', means channels are specified in
velocity.
        mode = 'frequency', means channels are specified in
frequency.
>>> mode expandable parameters (for modes other than 'mfs')
    Start, width are given in units of channels, frequency
or velocity as indicated by mode, but only channel
is complete.
    nchan -- Number of channels (planes) in output image
        default: 1; example: nchan=3
    start -- Start input channel (relative-0)
        default=0; example: start=5
    width -- Output channel width in units of the input
channel width (>1 indicates channel averaging)
        default=1; example: width=4
examples:
    spw = '0,1'; mode = 'mfs'
        will produce one image made from all channels in spw
0 and 1
    spw='0:5~28^2'; mode = 'mfs'
        will produce one image made with channels
(5,7,9,...,25,27)
    spw = '0'; mode = 'channel': nchan=3; start=5; width=4
        will produce an image with 3 output planes
        plane 1 contains data from channels (5+6+7+8)
        plane 2 contains data from channels (9+10+11+12)
        plane 3 contains data from channels (13+14+15+16)
    spw = '0:0~63^3'; mode='channel'; nchan=21; start = 0;
width = 1
        will produce an image with 20 output planes

```

```

    Plane 1 contains data from channel 0
    Plane 2 contains data from channel 2
    Plane 21 contains data from channel 61
    spw = '0:0~40^2'; mode = 'channel'; nchan = 3; start =
5; width = 4
    will produce an image with three output planes
    plane 1 contains channels (5,7)
    plane 2 contains channels (13,15)
    plane 3 contains channels (21,23)
psfmode -- method of PSF calculation to use during minor cycles:
    default: 'clark': Options: 'clark','hogbom'
    'clark' use smaller beam (faster, usually good enough)
    'hogbom' full-width of image (slower, better for poor
uv-coverage)
    Note: psfmode will be used to clean is imagermode = ''
imagermode -- Advanced imaging e.g mosaic or Cotton-Schwab clean
    default: imagermode='': Options: '', 'csclean', 'mosaic'
    default '' => psfmode cleaning algorithm used
>>> imagermode='mosaic' expandable parameter(s):
    Image as a mosaic of the different pointings (uses csclean
style too)
    mosweight -- Individually weight the fields of the mosaic
    default: False; example: mosweight=True
    This can be useful if some of your fields are more
sensitive than others (i.e. due to time spent
on-source); this parameter will give more weight to
higher sensitivity fields in the overlap regions.
    ftmachine -- Gridding method for the image;
    Options: ft (standard interferometric gridding), sd
(standard single dish) both (ft and sd as appropriate),
mosaic (gridding use PB as convolution function)
    default: 'mosaic'; example: ftmachine='ft'
    scaletype -- Controls scaling of pixels in the image plane.
    (Not fully implemented...for now only controls
what is seen if interactive=True...but in the future will
control the image on which clean components are searched)
    default='SAULT'; example: scaletype='PBCOR'
    Options: 'PBCOR','SAULT'
    'SAULT' when interactive=True shows the residual
with constant noise across the mosaic. If
pbcor=False, the final output image is NOT
corrected for the PB pattern, and therefore is
not "flux correct". Division of SAULT
<im名称>.image by the <im名称>.flux image
will produce a "flux correct image", can also

```

be achieved by setting `pbcor=True`.

`'PBCOR'` uses the SAULT scaling scheme for deconvolution, but if `interactive=True` shows the primary beam corrected image; the final PBCOR image is "flux correct" if `pbcor=True`.

>>> `imagermode='csclean'` expandable parameter(s): Image using the Cotton-Schwab algorithm in between major cycles

`cyclefactor` -- Change the threshold at which the deconvolution cycle will stop, `degrid` and subtract from the visibilities. For poor PSFs, reconcile often (`cyclefactor=4` or `5`); For good PSFs, use `cyclefactor 1.5` to `2.0`. Note: `threshold = cyclefactor * max sidelobe * max residual`.

default: `1.5`; example: `cyclefactor=4`

`cyclespeedup` -- Cycle threshold doubles in this number of iterations default: `-1`;
example: `cyclespeedup=3`

try `cyclespeedup = 50` to speed up cleaning

`multiscale` -- set of scales to use in deconvolution. If set, cleans with several resolutions using `hobgom clean`. The scale sizes are in units of `cellsize`. So if `cell='2arcsec'`, a `multiscale scale=10 = 20arcsec`. First scale should always be `0` (point), we suggest second on the order of synthesized beam, third 3-5 times synthesized beam, etc. For example if synthesized beam is `10"` and `cell="2"`, try `multscale = [0,5,15]`. Note, `multiscale` is currently a bit slow.

default: `multiscale=[]` (standard CLEAN using `psfmode` algorithm, no multi-scale). Example: `multscale = [0,5,15]`

>>> `multiscale` expandable parameter(s): `negcomponent` -- Stop component search when the largest scale has found this number of negative components; `-1` means continue component search even if the largest component is negative. default: `-1`; example: `negcomponent=50`

`imsize` -- Image pixel size (x,y)
default = `[256,256]`; example: `imsize=[350,350]`
`imsize = 500` is equivalent to `[500,500]`

`cell` -- Cell size (x,y)
default= `'1.0arcsec'`;
example: `cell=['0.5arcsec','0.5arcsec']` or
`cell=['1arcmin', '1arcmin']`
`cell = '1arcsec'` is equivalent to `['1arcsec','1arcsec']`
NOTE:`cell = 2.0 => ['2arcsec', '2arcsec']`

`phasecenter` -- direction measure or `fieldid` for the mosaic center

```

    default: '' => first field selected ; example: phasecenter=6
    or phasecenter='J2000 19h30m00 -40d00m00'
restfreq -- Specify rest frequency to use for output image
    default='' Occasionally it is necessary to set this (for
    example some VLA spectral line data). For example for
    NH_3 (1,1) put restfreq='23.694496GHz'
stokes -- Stokes parameters to image
    default='I'; example: stokes='IQUV';
    Options: 'I','IV','QU','IQUV','RR','LL','XX','YY','RLL','XXYY'
niter -- Maximum number iterations,
    if niter=0, then no CLEANing is done ("invert" only)
    default: 500; example: niter=5000
gain -- Loop gain for CLEANing
    default: 0.1; example: gain=0.5
threshold -- Flux level at which to stop CLEANing
    default: '0.0mJy';
    example: threshold='2.3mJy' (always include units)
           threshold = '0.0023Jy'
           threshold = '0.0023Jy/beam' (okay also)
interactive -- use interactive clean (with GUI viewer)
    default: interactive=False
    example: interactive=True
    interactive clean allows the user to build the cleaning
    mask interactively using the viewer. The viewer will
    appear every npercycle iteration, but modify as needed
    The final interactive mask is saved in the file
    imaname_interactive.mask. The initial masks use the
    union of mask and cleanbox (see below)
>>> interactive=True expandable parameter npercycle -- this is the
    number of iterations between each clean to update mask
    interactively. Set to about niter/5, but can also be
    changed interactively.
mask -- Specification of cleanbox(es), mask image(s), and/or
region(s) to be used for CLEANing. As long as the image has
the same shape (size), mask images from a previous
interactive session can be used for a new execution. NOTE:
the initial clean mask actually used is the union of what
is specified in mask and <imaname>.mask default: [] (no
masking); Possible specification types: (a) Explicit
cleanbox pixel ranges example: mask=[110,110,150,145] clean
region with blc=110,100; trc=150,145 (pixel values) (b)
Filename with cleanbox pixel values with ascii format:
example: mask='mycleanbox.txt' <fieldid blc-x blc-y
trc-x trc-y>; on each line
1 45 66 123 124

```

```

2 23 100 300 340
(c) Filename for image mask example: mask='myimage.mask'
(d) Filename for region specification (e.g. from viewer)
example: mask='myregion.rgn' (e) Combinations of any of the
above example: mask=[[110,110,150,145], 'mycleanbox.txt',
'myimage.mask', 'myregion.rgn']
    uvtaper -- Apply additional uv tapering of the visibilities.
              default: uvtaper=False; example: uvtaper=True
&gt;&gt;&gt; uvtaper=True expandable parameters
    outertaper -- uv-taper on outer baselines in uv-plane
                [bmaj, bmin, bpa] taper Gaussian scale in uv or
angular units. NOTE: uv taper in (klambda) is roughly on-sky
                FWHM(arcsec/200)
                default: outertaper=[]; no outer taper applied
example: outertaper=['5klambda'] circular taper
FWHM=5 kilo-lambda
                outertaper=['5klambda', '3klambda', '45.0deg']
                outertaper=['10arcsec'] on-sky FWHM 10"
                outertaper=['300.0'] default units are meters
in aperture plane
    innertaper -- uv-taper in center of uv-plane
                [bmaj,bmin,bpa] Gaussian scale at which taper falls to
zero at uv=0
                default: innertaper=[]; no inner taper applied
                NOT YET IMPLEMENTED
modelimage -- Name of model image(s) to initialize cleaning. If
multiple images, then these will be added together to
form initial staring model NOTE: these are in addition
to any initial model in the &lt;imagename&gt;.model image file
default: '' (none); example: modelimage='orion.model'
modelimage=['orion.model', 'sdorion.image'] Note: if the
units in the image are Jy/beam as in a single-dish
image, then it will be converted to Jy/pixel as in a
model image, using the restoring beam in the image
header
weighting -- Weighting to apply to visibilities:
              default='natural'; example: weighting='uniform';
Options: 'natural', 'uniform', 'briggs',
'superuniform', 'briggsabs', 'radial'
&gt;&gt;&gt; Weighting expandable parameters
For weighting='briggs' and 'briggsabs'
    robust -- Brigg's robustness parameter
              default=0.0; example: robust=0.5;
              Options: -2.0 to 2.0; -2 (uniform)/+2 (natural)
For weighting='briggsabs'

```



```

        noise    -- noise parameter to use for Briggs "abs"
weighting
        example noise='1.0mJy'
        For superuniform/briggs/briggsabs weighting
        npixels -- number of pixels to determine uv-cell size
        for weight calculation
        example npixels=7
restoringbeam -- Output Gaussian restoring beam for CLEAN image
        [bmaj, bmin, bpa] elliptical Gaussian restoring beam
        default units are in arc-seconds for bmaj,bmin, degrees
        for bpa default: restoringbeam=[]; Use PSF calculated
        from dirty beam.
example: restoringbeam=['10arcsec'] circular Gaussian
        FWHM 10" example:
        restoringbeam=['10.0','5.0','45.0deg'] 10"x5"
        at 45 degrees
pbcor -- Output primary beam-corrected image
        default: pbcor=False; output un-corrected image
example: pbcor=True; output pb-corrected image (masked outside
        minpb) Note: if you set pbcor=False, you can later
        recover the pbcor image by dividing by the .flux image
        (e.g. using immath)
minpb -- Minimum PB level to use default=0.1; example:
        minpb=0.01 Note: this minpb is always in effect
        (regardless of pbcor=True/False)
async -- Run asynchronously
        default = False; do not run asynchronously

```

```
</example>
```

```
</task>
```

```
</casaxml>
```

H.3.2 File task_clean.py

Task clean implementation file.

```

import os
from taskinit import *
from cleanhelper import *

```

```

def clean(vis,imagename,field, spw, selectdata, timerange, uvrange, antenna,
         scan, mode,niter, gain,threshold, psfmode, imagermode, ftmachine,
         mosweight, scaletype, multiscale, negcomponent,interactive,mask,
         nchan,start,width,imsize,cell, phasecenter, restfreq, stokes,
         weighting,robust, uvtaper,outertaper,innertaper, modelimage,
         restoringbeam,pbcor, minpb, noise, npixels, npercycle, cyclefactor,
         cyclespeedup):

#Python script

        casalog.origin('clean')

maskimage=''
if((mask==[]) or (mask=='')):
mask=['']
if (interactive):
if( (mask=='') or (mask==['']) or (mask==[])):
maskimage=imagename+'.mask'

#try:
if(1):
imCln=imtool.create()
imset=cleanhelper(imCln, vis)

if((len(imagename)==0) or (imagename.isspace())):
raise Exception, 'Cannot proceed with blank imagename'
casalog.origin('clean')

        imset.defineimages(imsize=imsize, cell=cell, stokes=stokes,
        mode=mode, spw=spw, nchan=nchan,
        start=start, width=width,
        restfreq=restfreq, field=field,
        phasecenter=phasecenter)

        imset.datselweightfilter(field=field, spw=spw,
timerange=timerange, uvrange=uvrange,
antenna=antenna, scan=scan,
wgtype=weighting, robust=robust,
noise=noise, npixels=npixels,
mosweight=mosweight,
innertaper=innertaper,
outertaper=outertaper)
if(maskimage==''):

```

```

maskimage=imagenam+'.mask'
imset.makemaskimage(outputmask=maskimage,imagenam=imagenam,
                    maskobject=mask)

###define clean alg
alg=psfmode
if(multiscale==[0]):
multiscale=[]
if((type(multiscale)==list) and (len(multiscale)>0)):
alg='multiscale'
imCln.setscales(scalemethod='uservector',
uservector=multiscale)
if(imagermode=='csclean'):
alg='mf'+alg
if(imagermode=='mosaic'):
if(alg.count('mf') <1):
alg='mf'+alg
imCln.setoptions(ftmachine=ftmachine, padding=1.0)
imCln.setvp(dovp=True)
###PBCOR or not
sclt='SAULT'
if((scaletype=='PBCOR') or (scaletype=='pbcor')):
sclt='NONE'
imCln.setvp(dovp=True)
else:
if(imagermode != 'mosaic'):
##make a pb for flux scale
imCln.setvp(dovp=True)
imCln.makeimage(type='pb', image=imagenam+'.flux')
imCln.setvp(dovp=False)
##restoring
imset.setrestoringbeam(restoringbeam)
###model image
imset.convertmodelimage(modelimages=modelimage,
outputmodel=imagenam+'.model')

####after all the mask shenanigans...make sure to use the
####last mask
maskimage=imset.outputmask
if((imagermode=='mosaic')):
imCln.setmfcontrol(stoplargenegatives=negcomponent,scaletype=sclt,
                    minpb=minpb,cyclefactor=cyclefactor,
                    cyclespeedup=cyclespeedup,

```

```

                                fluxscale=[imagenam+'.flux'])
else:
imCln.setmfcontrol(stoplargenegatives=negcomponent,
                   cyclefactor=cyclefactor, cyclespeedup=cyclespeedup)

imCln.clean(algorithm=alg,niter=niter,gain=gain,
            threshold=qa.quantity(threshold,'mJy'),
            model=[imagenam+'.model'],
            residual=[imagenam+'.residual'],
            image=[imagenam+'.image'],
            psfimage=[imagenam+'.psf'],
            mask=maskimage, interactive=interactive,
            npercycle=npercycle)

imCln.close()
presdir=os.path.realpath('.')
newimage=imagenam
if(imagenam.count('/') > 0):
newimage=os.path.basename(imagenam)
os.chdir(os.path.dirname(imagenam))
result      = '\'' + newimage + '.image' + '\'';
fluxscale_image = '\'' + newimage + '.flux' + '\'';
if (pbcor):
if(sclt != 'NONE'):
##otherwise its already divided
ia.open(newimage+'.image')

pixmask = fluxscale_image+'>'+str(minpb);
ia.calcmask(pixmask,asdefault=True);

pixels='iif('+ fluxscale_image+'>'+str(minpb)+','
                                + result+'/'+fluxscale_image+', 0)'

ia.calc(pixels=pixels)
ia.close()
else:
## people has imaged the fluxed corrected image
## but want the
## final image to be non-fluxed corrected
if(sclt=='NONE'):
ia.open(newimage+'.image')
result=newimage+'.image'
fluxscale_image=newimage+'.flux'
pixels=result+'*'+fluxscale_image
ia.calc(pixels=pixels)
ia.close()
os.chdir(presdir)

```

```
del imCln

# except Exception, instance:
# print '*** Error *** ',instance
# raise Exception, instance
```